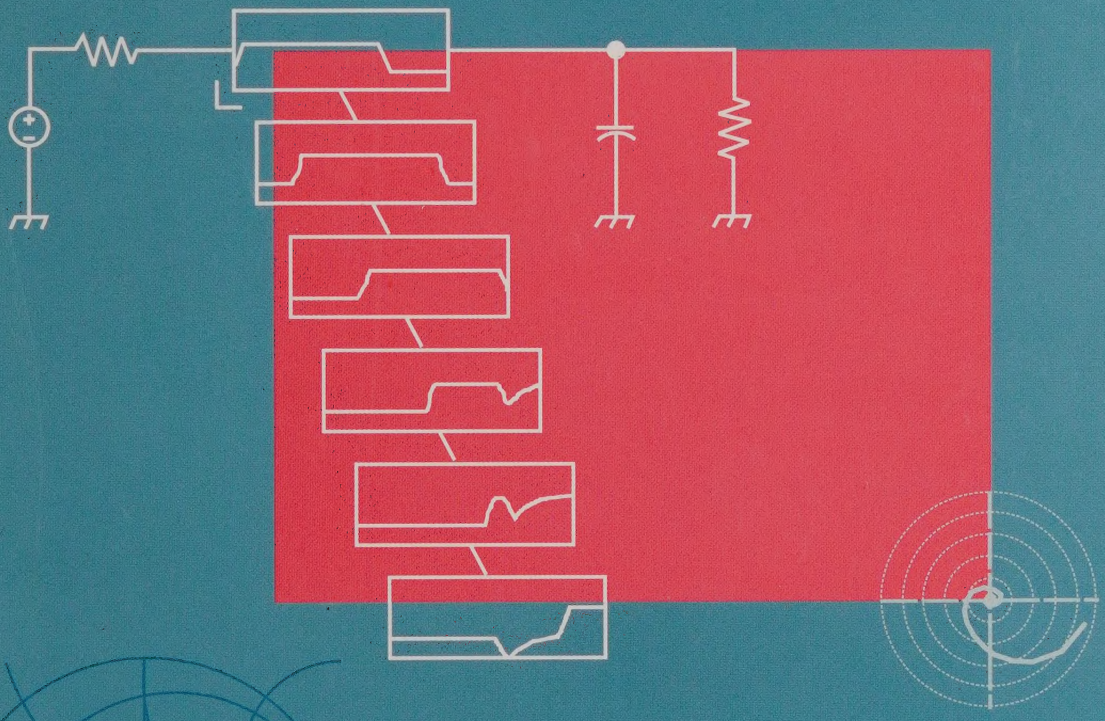
TRAVIS Student

Transmission Line Visualization Software and User's Manual

Student Version



Robert G. Kaires
Barton T. Hickman

Digitized by the Internet Archive
in 2025

https://archive.org/details/manualsplus_09013

To take advantage of big savings on future releases, mail your registration card today!

At Artech House, we are always looking for ways to make our current software offerings better, such as expanding capability or upgrading user features.

As a registered user of this software, you will be first on our list to receive notices of upgrades when they become available. More importantly, only registered users will be entitled to purchase future releases at substantial discounts.

Please take a few minutes to fill out the card below and mail it today.

Customers in Europe, Scandinavia, the Middle East, and Africa:

ARTECH HOUSE, INC.
Portland House
Stag Place
London, SW1E 5XA
UK

All other customers:

ARTECH HOUSE, INC.
685 Canton Street
Norwood, MA 02062
USA

TRAVIS Student: Transmission Line Visualization Software and User's Manual
Student Version

I have purchased this software and wish to be notified of future upgrades as soon as they become available.

Name _____

Title _____

Company _____

Street address _____

_____

Daytime telephone _____

E-mail address _____

TRAVIS Student

Transmission Line Visualization Software and User's Manual

Student Version

For a complete listing of the *Artech House Microwave Library*,
turn to the back of this manual.

TRAVIS Student

Transmission Line Visualization Software and User's Manual

Student Version

Robert G. Kaires and Barton T. Hickman

Artech House
Boston • London

Library of Congress Cataloging-in-Publication Data
Kaires, Robert G.
TRAVIS Student [computer file]: transmission line visualization software and user's manual/Robert G. Kaires and Barton Hickman.–Student version.
1 computer disk; 3 1/2 in. + 1 user's manual.
System requirements: IBM PC 80386 or later version or 100% compatible; minimum of 4MB RAM; Windows 3.1 or later version with 1MB free; hard disk with 5MB free; math coprocessor.
Title from user's manual.
Audience: Professional engineering students.
Summary: Electronic circuit simulation program which teaches about the nature of reflections/transmission of waves on transmission lines. Allows the user to analyze the voltage and current waveforms propagation within transmission lines.
ISBN 0-89006-838-0
1. Multiconductor transmission lines–Software. 2. Electronic circuits–Software. 3. Wave guides–Software. I. Hickman, Barton. II. Title.
TK7872.T74 <MRCRR> 95-5053
621.3815–dc12 CIP

© 1996 ARTECH HOUSE, INC.
685 Canton Street
Norwood, MA 02062

All rights reserved. Printed and bound in the United States of America. No part of this manual may be reproduced or utilized in any form or by any means, electronic or mechanical, including photocopying, recording, or by any information storage and retrieval system, without permission in writing from the publisher. All terms mentioned in this manual that are known to be trademarks or service marks have been appropriately capitalized. Artech House cannot attest to the accuracy of this information. Use of a term in this manual should not be regarded as affecting the validity of any trademark or service mark.

International Standard Book Number: 0-89006-838-0
Library of Congress Catalog Card Number: 95-5053

10 9 8 7 6 5 4 3 2 1

Table of Contents

Disclaimer of Warranty vii
Introduction ix
1 Installation 1
2 Quick Start 3
3 Entering and Editing a Circuit 5
3.1 File Menu 5
3.2 Edit Menu 10
3.3 Place Menu 17
3.4 Placing Components 17
3.5 Placing Wires 18
3.6 Placing Grounds, Node Markers, Terminals, and Ports 19
3.7 Editing Component Parameters 20
3.8 Mathematical Expressions 23
3.9 Variables 24
3.10 Primitive Components 25
3.11 Editing the Circuit 34
3.12 Subcircuits 34
3.13 Units 42
3.14 Transmission Line Designer 42
4 Preparing for and Running the Analysis 49
4.1 DC Analysis 49
4.2 AC Analysis 49
4.3 Port Analysis 50
4.4 Transient Analysis 51
4.5 Analysis Menu 51
5 Seeing Analysis Results 59
5.1 DC Analysis Results 59
5.2 AC Analysis Results 59
5.3 Port Analysis Results 60
5.4 TRansient Analysis Results 60
5.5 Transmission Line Analysis Results (Animation) 61
5.6 Waveform Math 63
5.7 See Results Menu 64

5.8 Plot Windows 81
Error Messages 89
Glossary 95
Index 101

Disclaimer of Warranty

The technical descriptions, procedures, and computer programs featured herein have been developed with the greatest care and have been useful to the authors in a broad range of applications; however, they are provided as is, without warranty of any kind. ARTECH HOUSE, Inc., and the authors of TRAVIS Pro (Transmission Line Visualization Software and User's Manual--Professional Version) and TRAVIS Student (Transmission Line Visualization Software and User's Manual--Student Version) make no warranties, expressed or implied, that the equations, programs, and procedures in the software and manuals are free of error, or are consistent with any particular standard of merchantability, or will meet your requirements for any particular application. They should not be relied on for solving a problem whose incorrect solution could result in injury to a person or loss of property. Any use of the programs or procedures in such a manner is at the user's own risk. The authors and publisher disclaim all liability for direct, incidental, or consequential damages resulting from your use of the programs or procedures described herein.

Introduction

TRAVIS is an electronic circuit simulation program designed primarily to allow visualization of propagating waveforms as they travel along transmission lines. TRAVIS does this in the format of a more general circuit simulator program to allow full flexibility and greater ease in creating circuit analyses that contain the transmission lines of interest.

ACKNOWLEDGMENTS

We would like to acknowledge those responsible for the sparse matrix solving routines used by TRAVIS:

The University of California at Berkeley

Author:
Kenneth S. Kundert

Advising professor:
Alberto Sangiovanni-Vincentelli

Copyright © 1985,1986,1987,1988
by Kenneth S. Kundert and the University of California.

SYSTEM REQUIREMENTS

-MS Windows 3.1.
-2 Mbytes free hard disk space.
-1 Mbyte free memory from within Windows (i.e., after Windows and any drivers have taken what they need). A machine with 4 Mbytes of free memory generally can handle this.
-80386 or newer processor and corresponding math coprocessor.
-EGA or greater resolution.

-2- or 3- button Mouse.

(Note: The program was developed on an 80486/33 with 8 Mbytes of memory and runs nicely on such a system.)

LIMITS OF TRAVIS

There are some basic limits to the size and complexity of circuits that TRAVIS can deal with. All of these limits assume that you do not run out of memory space in Windows first. The limits of the different parts of the program are described below.

TRAVIS 1.0 Professional has the following limits:

-500 subcircuits in the subcircuit library.
-10,000 primitive components within the entire project netlist.
-10,000 transmission lines within the entire project netlist.
-200 nodes within the entire project netlist.
-20 layers in the project hierarchy.
-1,000 components within any one subcircuit (or the main circuit).
-15,000 wire segments within any one subcircuit (or the main circuit).

TRAVIS 1.0 Student has the following limits.

-5 subcircuits in the subcircuit library
-20 primitive components within the entire project netlist.
-3 transmission lines within the entire project netlist.
-10 nodes within the entire project netlist.
-2 layers in the project hierarchy.
-1,000 components within any one subcircuit (or the main circuit).
-15,000 wire segments within any one subcircuit (or the main circuit).

TRAVIS FILES

TRAVIS creates various temporary files that contain circuit or analysis information. TRAVIS does not add any files to any system directories nor does it modify any system files (such as win.ini or system.ini). All files that TRAVIS does create (besides files that you explicitly create) are kept in the local TRAVIS directory.

Analysis files

When analyses are run, TRAVIS creates several files to contain the various types of analysis information (dc voltages, animation plots, two-port analysis, etc). These files

have coded names that will be unique for a given TRAVIS session. The names of these files are listed below, along with the corresponding contents.

z??????.f00	Ac analysis node voltages.
z??????.g00	Ac analysis source currents.
z??????.c00	Ac analysis animation plot signals.
z??????.t00	Transient analysis node voltages.
z??????.u00	Transient analysis source currents.
z??????.a00	Transient analysis animation plot signals.
z??????.bd1	dc analysis node voltages.
z??????.id1	dc analysis source currents.
z??????.f01	Port 1 analysis voltage info.
z??????.g01	Port 1 analysis current info.
z??????.f02	Port 2 analysis voltage info.
z??????.g02	Port 2 analysis current info.

The "??????" represents a unique name that is chosen each time TRAVIS is run. When TRAVIS is terminated, these files are deleted. If Windows or TRAVIS crashes while TRAVIS is running, these files will be left on disk, and you may want to manually delete them. These files are created in the same directory that TRAVIS is executing from (i.e., wherever TRAVIS.exe resides). Only the files that are necessary are created and saved--e.g., if you never run an ac analysis, then the first three files listed above are never created.

When you select "Save Subcircuit" or "Save Subcircuit As", just the one file whose name you choose is saved: <circuit name>.cir. None of the analysis results are saved, although they are still retained in the temporary files listed above until you exit TRAVIS.

When you select "Save Project" or "Save Project As", the file name you choose is created and all circuit, subcircuit, and analysis setup information is stored in that file. In addition, the files listed above are copied to the same root filename, but with their original extension.

Settings information file--travis.ini

The first time you run it, TRAVIS also creates a file called "travis.ini" in the local TRAVIS directory. This file contains settings information. If you delete travis.ini, TRAVIS will rebuild a default travis.ini the next time you run it. The default file listing and its meaning are listed below:

File listing:

```
---------------------------
[features]
undoredo=on

[warnings]
loseresults=off
```

"undoredo=<on/off>": Controls whether the "Undo/Redo" feature is enabled. Set to undoredo=off to disable this feature. On slower machines, the overhead involved in remembering every editing operation for possible undoing or redoing may slow down editing too much to suit your taste. Disabling this feature also saves memory.

"loseresults=<on/off>": If they exist, analysis results are always flushed when an editing operation is performed on the circuit. If you set loseresults=on, you will get a warning message whenever you are about to lose analysis results due to an attempted editing operation. By default, there is no warning.

Group file

This is the "travis.grp" file created in the installation directory for the TRAVIS program group.

Transmission line designer initialization file

The Transmission Line Designer tool has its own initialization file called "travline.ini". This file contains the basic setup information for this program. TRAVIS uses this file to transfer information between the Transmission Line Designer and the TRAVIS circuit editor.

Chapter 1

Installation

1. Turn on your computer and run MS Windows 3.1.
2. Insert the installation diskette into the floppy drive.
3. From Windows, execute the program "install.exe" on the installation disk. This is done by pulling down the "File" menu in the Windows Program Manager. A dialog box pops up prompting for a file name. Type "a:\install.exe" and click the "OK" button (assuming the installation floppy is in drive a).
4. The TRAVIS installation program pops up a dialog box showing the default installation directory (usually c:\travis). If you want TRAVIS installed in a different directory, edit the "Installation Directory" field.
5. Click the "OK" button. When the program is finished, it pops up a message box telling you it was successful. Click the "OK" button in this message box as well.
6. The installation program will attempt to create a TRAVIS program group and add a TRAVIS icon to that group.
7. Double-click the TRAVIS icon to start the program. (If for some reason the TRAVIS icon did not appear, then you will have to create your own group and icon pointing to the travis.exe file.)

Chapter 2

Quick Start

The following step-by-step procedure will show you how to load a project from disk, run a transient analysis, and view the transient and animation results of that analysis.

1. Start TRAVIS.
2. Pull down the "File" menu and select "Open Project".
3. The Open File dialog box appears. Select the file "example1.pro" and click the "OK" button.
4. Pull down the "Analysis" menu and select "Run Transient Analysis".
5. The Analysis Status dialog box pops up showing the progress of the analysis. When the analysis is complete, click the "OK" button.
6. Position the mouse pointer over the wire connecting to the right-hand side of the transmission line in the circuit (it has the node marker "out" attached to it).
7. Push down the left mouse button. A small menu pops up; select "v(out)" and release the mouse button.
8. A transient plot pops up with the voltage waveform from that wire. Notice how distorted the waveform looks due to reflections on the transmission line.
9. Pull down the "See Results" menu and select "Stick All Plots on Top". This will cause the plot to stick on top of the schematic. Pull down the "See Results" menu again and select "Use Active Plot". This will cause any more waveforms that you plot to be sent to the waveform plot that is already open.
10. Move and resize the plot window so that it does not obscure the circuit in the circuit window.
11. Pull down the "See Results" menu and select "Animate Transient Animation Plots". The Animation Control Dialog box pops up.
12. Press the "Play" button in the Animation Control dialog box. The voltage waveform within the transmission line starts animating. Also notice that an "X" marker

is moving along the waveform in the plot window corresponding to voltage versus time on that node.

13. When you are finished looking at the animation results, close the Animation Control dialog box by clicking the "Done" button.
14. Position the mouse pointer over the circuit window and click the right mouse button until the status text in the upper left corner of the circuit window says "Edit Circuit".
15. Position the mouse pointer over the resistor "R2" in the circuit and click the left mouse button. The Edit Component dialog box pops up for the resistor.
16. In the parameter field, the resistor parameters are listed, among them "r=1k". Change this to "r=50" and click the "OK" button of the dialog box. This will properly terminate the transmission line.
17. Repeat steps 4 through 7 above.
18. Now there are two waveforms in the plot.
19. Pull down the "See Results" menu and select "Animate Transient Animation Plots" again. The Animation Control dialog box pops up.
20. Press the "Play" button again. Now notice that the waveform has no reflections. Correlate this with what you see in the plot window.

Chapter 3

Entering and Editing a Circuit

There are three menus associated with Editing circuits: 1) File, 2) Edit, and 3) Place. The "File" menu contains items for saving or loading circuits from disk and sending them to the printer or a documentation application. The "Edit" menu contains items for modifying or deleting existing circuitry. The "Place" menu contains items for adding component and wires to the circuit. These three menus are described in the sections below.

3.1 FILE MENU

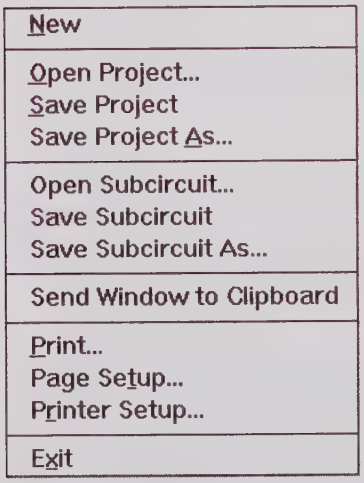


Fig. 3.1.1: File menu

The "File" menu items (of the circuit window) are described below.

3.1.1 New

Select the "New" item under the "File" menu of the main circuit window to clear out the current project. This also clears out the TRAVIS clipboard.

3.1.2 Open Project

To load a project from disk, select the "Open Project" item from the "File" menu of the main circuit window. A standard Windows Open File dialog box will pop up showing all files with a ".pro" suffix. These files contain TRAVIS projects. Either select the file you are looking for (by clicking on it with the left mouse button) or type it in. Then click the "OK" button. Click the "Cancel" button to abort this operation completely.

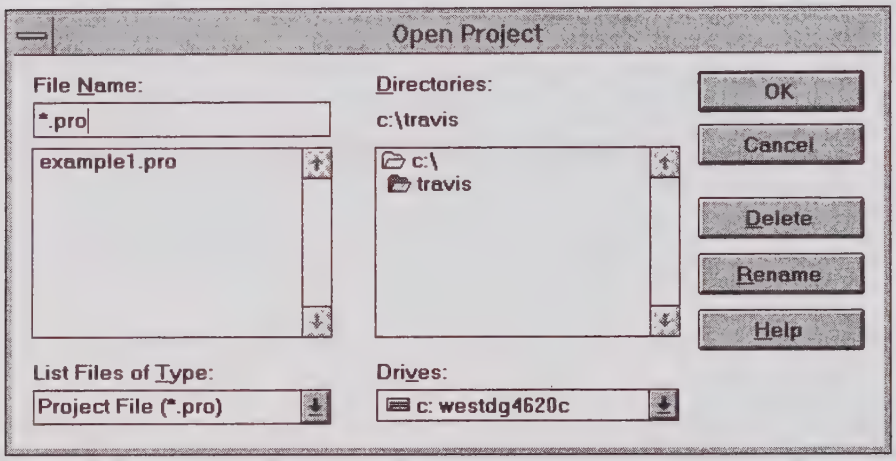


Fig. 3.1.2: Open Project dialog box

When you load a project, the project you are currently editing is replaced with the new project. This means that all previous analysis parameters and results as well as the main circuit and circuit body will be replaced. The analysis results associated with the new project are loaded into memory and are made available for viewing when you select the appropriate "See Results" menu item.

Delete and Rename buttons

To conveniently delete project files, you can select the file you want to delete, then click the "Delete" button. The file is deleted from disk. For project files (files with the ".pro" extension), all of the simulation results files are deleted also.

To conveniently rename project files, select the file you want to rename, then click the "Rename" button. Another File dialog box pops up asking you to enter the new name of the file. When you select the "OK" button in this second dialog box, the file is renamed. For project files (files with the ".pro" extension), all of the simulation results files are renamed as well.

The "Delete" and "Rename" features are included primarily so that if you want to rename a project, you can do so without having to also rename the many simulation re-

sults files that go along with it. (You can, however, use these features to delete or rename any files.)

3.1.3 Save Project

To save the current project to disk, select the "Save Project" item from the "File" menu of the main circuit window. The entire project will be saved to disk under the current project name (with the ".pro" extension). This means that all circuits, subcircuits, analysis settings, and last analysis results are saved. This contrasts with the "Save Subcircuit" menu item that will save only the circuit hierarchy to disk, not the analysis data or setup information.

3.1.4 Save Project As

To save the current project to disk under a new name, select the "Save Project As" item from the "File" menu of the main circuit window. A Save Project dialog box will pop up asking you to name the project. Enter or select the name of the project and click the "OK" button. The entire project will then be saved to disk. All project files must have a ".pro" extension; if you do not specify one, it will be added by TRAVIS. Click the "Cancel" button to abort this operation entirely.

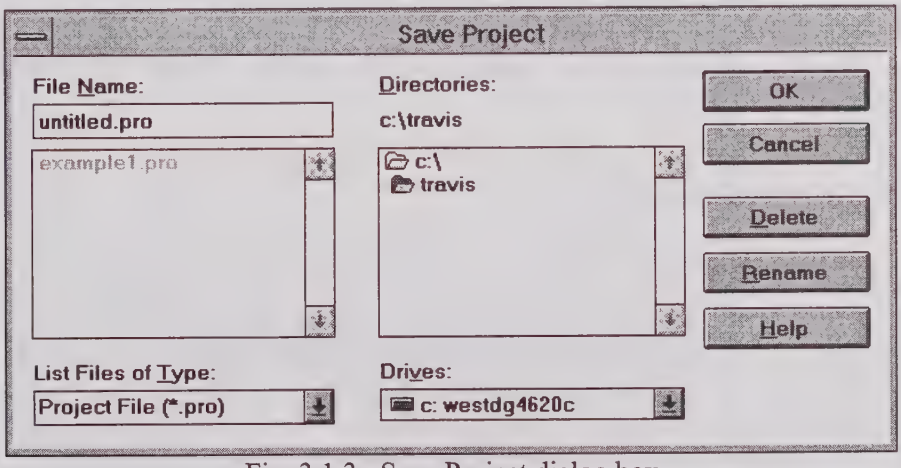


Fig. 3.1.3: Save Project dialog box

See also information about the "Delete" and "Rename" buttons in the "Open Project" menu item description.

3.1.5 Open Subcircuit

To load a subcircuit from disk, select the "Open Subcircuit" item from the "File" menu of the main circuit window. An Open File dialog box will pop up showing all files with

a ".cir" suffix. These files contain subcircuits; i.e., there is no analysis setup information or data in subcircuit files. Enter or select the subcircuit you want to open and click the "OK" button. The selected subcircuit will replace the main circuit in the current project or subcircuit that you are editing, but the analysis setup information remains unaffected. See "Subcircuits" for more information.

See also information about the "Delete" and "Rename" buttons in the "Open Project" menu item description.

3.1.6 Save Subcircuit

To save the current subcircuit to disk, select the "Save Subcircuit" item from the "File" menu of the main circuit window. The main circuit will be saved to disk as a subcircuit under the current subcircuit name, but no analysis results are saved.

3.1.7 Save Subcircuit As

To save the current subcircuit to disk under a new name, select the "Save Subcircuit As" item from the "File" menu of the main circuit window. A File Save dialog box will pop up asking you for the name of the subcircuit. Enter or select the name you want and click the "OK" button. You must use the file extension ".cir" when saving a subcircuit; if this is omitted, TRAVIS will add it. Click the "Cancel" button to abort this operation entirely. See "Subcircuits" for more information.

See also information about the "Delete" and "Rename" buttons in the "Open Project" menu item description.

3.1.8 Send Window to Clipboard

To send a bitmap picture of the circuit window to the Windows clipboard, select the "Send Window to Clipboard" item from the "File" menu of the circuit window. The circuit background color is automatically converted to white before being sent.

This feature is useful for exporting portions of your circuit to be included in a document being edited in a Windows word processor or other application. Simply select this menu item, then go to the other application and select that application's "Paste" function.

3.1.9 Print

To print the circuit window, select the "Print" item from the "File" menu of the circuit window. The circuit will be sent to the printer using the current printer and page settings.

3.1.10 Page Setup

To set up how the circuit window will look when printed, select the "Page Setup" item from the "File" menu of the circuit window.

A scaled representation of the printer page pops up showing how the circuit will look when printed. Selecting "Zoom in" or "Zoom out" from the "Change Scaling" menu adjusts the magnification of the circuit diagram. The scroll bars can be used to adjust the positioning of the circuit on the page.

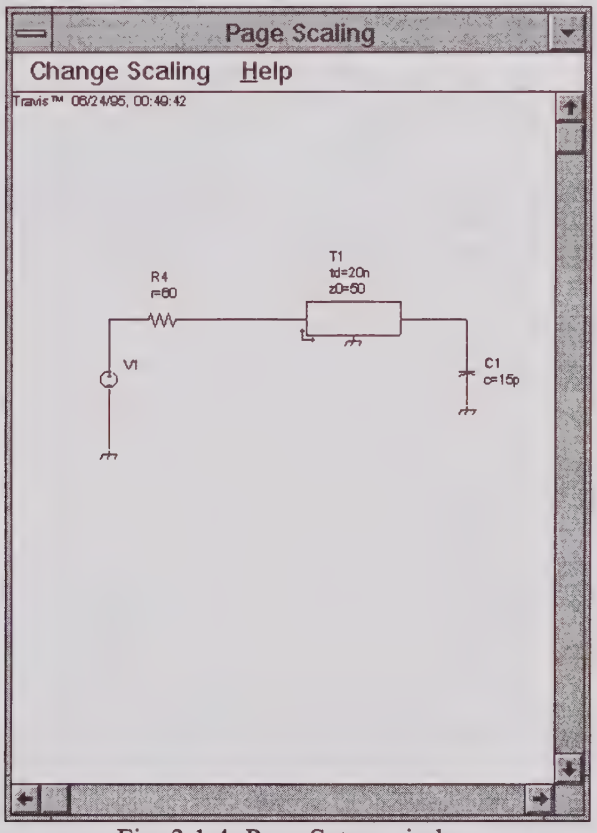


Fig. 3.1.4: Page Setup window

When you resize this window, it automatically readjusts itself to correspond to the aspect ratio of the printed page. You should set up the printer using the "Printer Setup" menu item before selecting this menu item since selecting and setting up the printer affects the page aspect ratio and, in general, how the circuit will look on the printed page.

To quit from this window and save the current page setup, just close the window by double clicking the upper left "System Menu" button of the window.

3.1.11 Printer Setup

To control the printer settings, select the "Printer Setup" item from the "File" menu of the circuit window.

When you select this menu item, the standard Printer Setup dialog box for your printer will pop up asking for various pieces of information such as the resolution and page orientation (and any other settings your particular printer uses).

The settings you select here are kept local to TRAVIS and will affect neither the system printer settings (in the Windows control panel) nor the printer settings of other programs you may currently have running.

3.1.12 Exit

Ahhh! It's been a long, hard day, and now it's time to quit TRAVIS. Select the "Exit" item from the "File" menu of the main circuit window to exit from TRAVIS. Be sure to save your project (or subcircuit as the case may be) before clicking "Yes" in the Exit Warning dialog box.

3.2 EDIT MENU

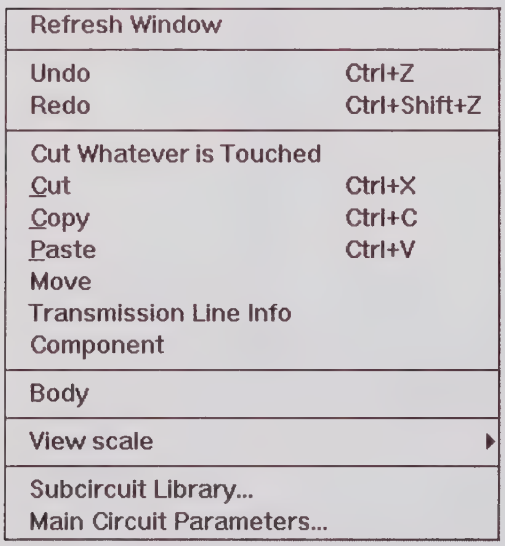


Fig. 3.2.1: Edit menu

The "Edit" menu items (of the circuit window) are described below.

3.2.1 Refresh Window

Sometimes during the course of editing the circuit, the editor leaves behind various graphical bits and pieces of components or text on the circuit. In order to clean up these bits and pieces, select the "Refresh Window" item from the "Edit" menu.

3.2.2 Undo

In order to undo an editing action, select the "Undo" menu item from the "Edit" menu. The most recent editing action will be undone.

You can undo up to 50 previous editing actions (cut, cut whatever is touched, paste, move, edit component). A "Move" command is treated as a separate cut and paste and so will require two "Undo's" to be undone.

The "Undo/Redo" feature does require some overhead during editing operations and thus slows them down a bit. If this becomes bothersome (on slower machines), you can disable this feature by editing the "travis.ini" file in the local TRAVIS directory. First close TRAVIS, then open "travis.ini" with a text editor and go to the line that contains "undoredo=on". Change it to "undoredo=off". For machines at or above a 486/33 in performance, there is little noticeable change in performance.

The "Undo" queue is not saved when you save the subcircuit or project to disk or when you exit TRAVIS.

3.2.3 Redo

In order to redo an editing action that was previously undone with the "Undo" menu item, select the "Redo" menu item from the "Edit" menu. The most recently undone editing action will be redone.

You can redo up to 50 previously undone editing actions (cut, cut whatever is touched, paste, move, edit component). A "Move" command is treated as a separate cut and paste and so will require two "Redo's" to be redone.

The "Undo/Redo" feature does require some overhead during editing operations and thus slows them down. If this becomes bothersome (on slower machines), you can disable this feature by editing the "travis.ini" file in the local TRAVIS directory. First close TRAVIS, then open "travis.ini" with a text editor and go to the line that contains "undoredo=on". Change it to "undoredo=off".

Whenever an editing action is performed, the "Redo" queue is flushed.

3.2.4 Cut Whatever Is Touched

Select the "Cut Whatever Is Touched" menu item from the "Edit" menu when you want to delete components or wires from the circuit one at a time at the click of the left mouse button.

When you select this menu item, the status text will display "Delete whatever is clicked on." Now whenever you click the left mouse button on a wire or component, it will be deleted. Items deleted in this fashion are not sent to the TRAVIS clipboard--therefore you can not subsequently paste them back down--but they are remembered by the "Undo" queue.

3.2.5 Cut

In order to delete the selected portion of the circuit, select the "Cut" menu item from the "Edit" menu. See "Editing the Circuit" for more detailed information on selecting parts of the circuit for the purpose of cutting, copying, and moving.

When you cut components, they are not permanently deleted--they go to the TRAVIS clipboard and may be pasted back down on the circuit. The cut circuitry may also be pasted into another subcircuit in the project by simply going into the open circuit window of that subcircuit and selecting the "Paste" menu item from the "Edit" menu of that circuit window.

3.2.6 Copy

In order to copy the selected portion of the circuit (for the purpose of pasting a copy of it elsewhere in the project or subcircuit), select the "Copy" menu item from the "Edit" menu. A copy of the selected circuitry is sent to the TRAVIS clipboard. See "Editing the Circuit" for more detailed information on selecting parts of the circuit for the purpose of copying (as well as other editing operations).

The "Copy" menu selection is similar to the "cut" menu selection except that the highlighted circuitry is not deleted.

3.2.7 Paste

Selecting the "Paste" menu item from the "Edit" menu causes previously copied or cut portions of circuitry to be pasted on the circuit. A copy of circuitry previously cut or copied appears attached to the mouse pointer.

Move the mouse pointer to the spot you want to place the circuitry, and push down on the left mouse button. A small menu pops up at the mouse location with three selections: 1) Place, 2) Rotate, and 3) Mirror. The menu pops up with the "Place" item already selected. Releasing the left mouse button on this selection causes a copy of the circuitry to be placed on the circuit. The circuitry remains attached to the mouse pointer so that you can continue to place copies of it in other locations of the circuit.

Releasing the mouse button on the "Rotate" selection causes the circuitry to rotate 90 degrees clockwise. The "Mirror" selection causes it to be horizontally mirrored (i.e., mirrored about a vertical axis through the center of the circuitry.) By using combinations of "Rotate" and "Mirror", any orientation can be achieved.

Note that if you just click (push down and release) the left mouse button, the circuitry will be placed on the circuit.

To cancel further placement of the circuitry, click on the right mouse button. The circuitry that was attached to the mouse pointer disappears and the editor returns to the normal edit circuit mode.

Note: Once you have copied or cut a portion of circuitry from a circuit window, you can paste it into any other circuit or subcircuit window of that project by selecting the "Paste" menu item of that window.

3.2.8 Move

If you want just want to move a selected piece of the circuit, select the "Move" menu item from the "Edit" menu. The selected circuitry becomes attached to the mouse pointer. You can then move, rotate, or mirror the circuit section before placing it, as described under "Paste" in the "Edit" menu. A "Move" operation is equivalent to a simultaneous cut and paste; a copy of the moved circuitry is put into the TRAVIS clipboard.

3.2.9 Transmission Line Info

TRAVIS contains a Transmission Line Designer tool that allows you to see a graphical representation of a transmission line's cross section. The Transmission Line Designer also features the ability to calculate any transmission line parameter given the others.

In order to do this for a given transmission line, you must first select the "Transmission Line Info" menu item from the "Edit" menu of the circuit window. The status text will display "Click t-line for T-Line Designer." While in this mode, click the left mouse button over the transmission line whose cross section you want to examine. The Transmission Line Designer tool is popped up and filled in with that transmission line's characteristic impedance parameter. When you exit out of the Transmission Line Designer, any updated Zo value is fed back into the transmission line component. (See "Transmission Line Designer" for more information on using the tool.)

3.2.10 Body

If you plan on creating a subcircuit out of the circuit you are editing, you will want to create a subcircuit body to represent your subcircuit. To edit or create this body, select the "Body" menu item from the "Edit" menu of the main circuit window (this menu item is not available in subcircuit windows).

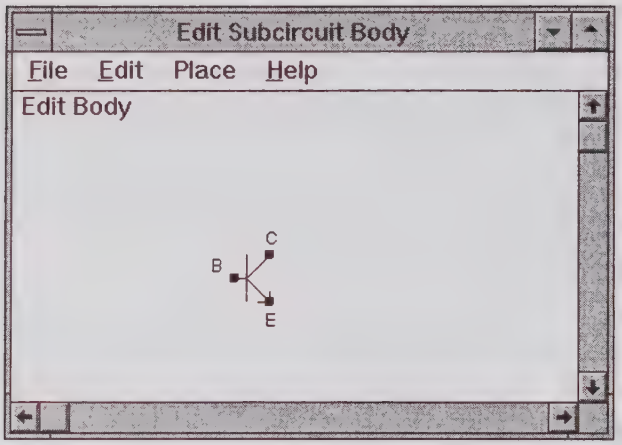


Fig. 3.2.2: Edit Subcircuit Body window

This will pop up the Edit Subcircuit Body window. The body corresponds to the main circuit in the project or circuit you are currently editing. By drawing a body to represent this circuit, you are enabling this circuit to be later pasted into other circuits as a subcircuit. (Actually, if you don't draw a body and save the current main circuit as a subcircuit, a default box-shaped body is automatically created and saved.) See "Subcircuits" for more information on editing the subcircuit body.

3.2.11 View Scale

Depending on your computer monitor, video mode, or eyesight, you might want to zoom in or zoom out on the circuit to see things more clearly. Selecting the "View Scale" item from the "Edit" menu of the circuit window causes a submenu with several zoom scale factors to pop up.

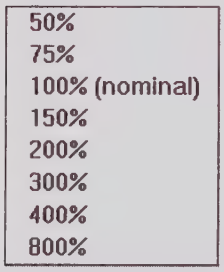


Fig. 3.2.3: View Scale submenu

The scaling selections range from 50% to 800%. 100% is the so-called nominal scaling and is what the program starts up in. Selecting less than 100% will cause the drawing to appear smaller (zoom out), resulting in more of the drawing being visible but with less detail. Selecting more than 100% will cause the drawing to appear larger (zoom in), resulting in less of the drawing being visible but with more detail.

Increasing the view scale is also useful in viewing animation plots more closely. This will not do just a graphical zoom of the plot--it will actually take advantage of the increased plot resolution to show more detail in the waveforms. For more information on animation plots, see the descriptions of the "Animate Transient Animation Plots" and "Animate AC Animation Plots" items under the "See Results" menu.

3.2.12 Subcircuit Library

To gain access to the project's subcircuit library, select the "Subcircuit Library" item from the "Edit" menu.

Selecting this menu item pops up the Subcircuit Library dialog box. For detailed information on the Subcircuit Library, see "Subcircuit Library."

3.2.13 Main Circuit Parameters/Subcircuit Parameters

When you want to edit the component parameters of a primitive or subcircuit component in your circuit, you simply click the left mouse button on that component and the Edit Component dialog box pops up allowing you to edit the parameters.

However, what if you want to edit the component information of the circuit or subcircuit you are currently editing? Simply select the "Main Circuit Parameters" menu item if you are editing the main circuit or the "Subcircuit Parameters" menu item if you are editing a subcircuit.

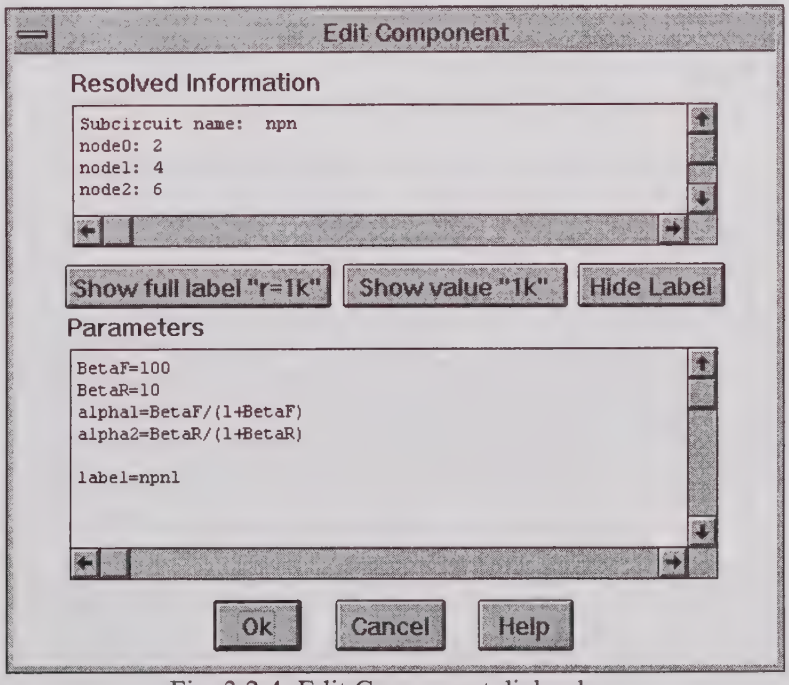


Fig. 3.2.4: Edit Component dialog box

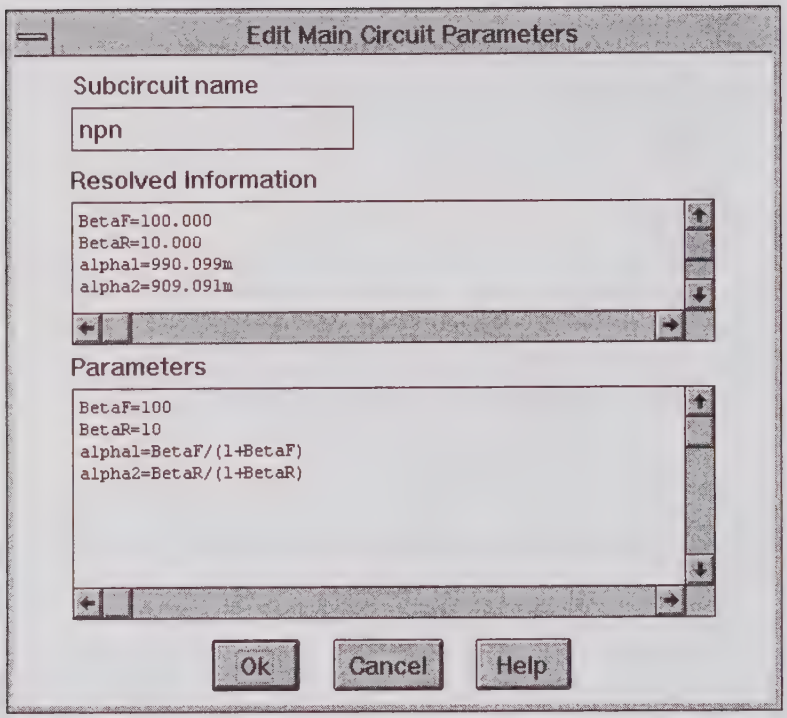


Fig. 3.2.5: Edit Main Circuit Parameters dialog box

If you are editing a subcircuit, then the Edit Component dialog box pops up (see Fig. 3.2.4)--the same one that you would get if you had clicked on the subcircuit component itself. Editing the parameters here is equivalent to if you had clicked on the component itself one level up in the circuit hierarchy (see "Editing Component Parameters" for more information).

If you are editing the main circuit, then the Edit Main Circuit Parameters dialog box pops up (Fig. 3.2.5). There are two differences between this dialog box and the Edit Component dialog box: 1) You can't control visibility of labels; this function is not applicable here, and 2) You can change the name of the main circuit. This name is not the name of the project; it is the name that the subcircuit will have when the project is saved as a subcircuit (see "Save Subcircuit"). In addition, the parameters that you enter into the parameter field will become the default parameters of the subcircuit component when it is loaded and placed into another project.

Note: Because TRAVIS uses the subcircuit name as the subcircuit filename (<subcircuit_name>.cir), all subcircuit names must be no longer than eight characters.

3.3 PLACE MENU

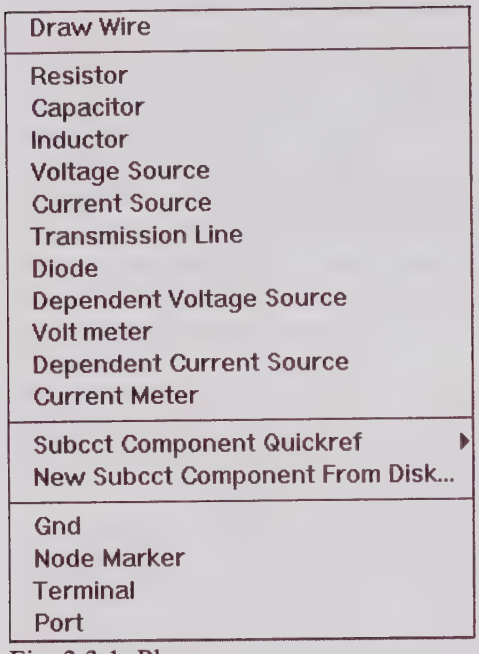


Fig. 3.3.1: Place menu

See "Placing Components" and "Editing Component Parameters" for specific information about what happens when these menu items are selected and how to edit the components once they are placed.

3.4 PLACING COMPONENTS

TRAVIS has two types of components: subcircuit components and primitive components. Primitive components are the fundamental components that are hardwired into TRAVIS. All circuits are constructed from primitive components. All of the primitive components can be found on the "Place" menu of the circuit window. (They are in the second grouping of this menu right after the "Draw Wire" item.)

Subcircuit components represent a subcircuit that may be constructed from primitive components and other subcircuit components. A circuit hierarchy is formed when subcircuits are used.

To place a component:

1. Pull down the "Place" menu and select one of the components in the menu. A copy of the component will be attached to the mouse pointer.
2. Move the mouse pointer to the spot where you want to place the component.

3. Push down on the left mouse button. A small menu pops up at the mouse location with three selections: 1) Place, 2) Rotate, and 3) Mirror. The menu pops up with the "Place" item already selected.

4. Release the left mouse button on the "Place" selection. A copy of the component is placed on the circuit. The component remains attached to the mouse pointer so that you can continue to place copies of it in other locations of the circuit. Releasing the mouse button on the "Rotate" selection causes the component to rotate 90 degrees clockwise. The "Mirror" selection causes the component to be horizontally mirrored (i.e., mirrored about a vertical axis through the center of the component). By using combinations of "Rotate" and "Mirror", any orientation can be achieved.

To place a difference type of component, simply pull down the "Place" menu again and select a different component (step 1.)

Note that if you just click (push down and release) the left mouse button, the component will be placed on the circuit.

When a component is placed on the circuit, it is automatically connected to any other components, terminals, wires, etc., that its terminals coincide with.

To cancel further placement of the component, click on the right mouse button. The component that was attached to the mouse pointer disappears and the editor returns to the edit circuit mode.

Placing subcircuit components is the same as placing primitive components once you select one and get it attached to the mouse pointer. See "Placing Subcircuit Components" for more information.

3.5 PLACING WIRES

To add a wire to the circuit:

1. Pull down the "Place" menu and select "Draw Wire" (the first selection in the menu.) The status text will display "Click to start wire".

2. Move the mouse pointer to the location on the circuit where you want to start drawing the wire and click on the left mouse button. Now the status text will say "Click to end wire".

3. You'll now notice that as you move the mouse, a wire is being drawn from the location that you clicked the mouse button. Also notice that the wire snaps to either vertical or horizontal orientation. To complete placement of the wire, click on the left button again. The wire segment is placed, and you are now drawing another wire segment.

4. Click the right mouse button to stop placement of the wire. Clicking the right button again takes you completely out of "Draw wire" mode and the status text will again say "Edit Circuit".

As wires are placed, they are automatically connected to existing circuitry. Two wires that cross each other will not connect.

3.6 PLACING GROUNDS, NODE MARKERS, TERMINALS, AND PORTS

TRAVIS has four types of node markers. All four of these are at the bottom of the "Place" menu. The mechanics of placing node markers is the same as placing components; see "Placing Components" for more information.

The property that node markers have in common is that they distinguish the node they are connected to for a particular purpose. These purposes are described below.

Node marker

This is the generic node marker. A node marker forces a node to the name of the node marker itself. As such, two node markers with the same name placed on different nodes will cause those nodes to be shorted together.

Another important use of a node marker is simply to label a node with a name you choose. Nodes are automatically numbered by TRAVIS when an analysis is run or when the netlist is written out. You can see what these node numbers are by pushing down the left mouse button on a wire (to see the node number of that wire) or by reading the resolved information field of the Edit Component dialog box (to see the node numbers of that component's pins). However, if you want a particular node to be named something specific (e.g., "out"), just place a node marker on that node and change its label to "out" (i.e., open the Edit Component dialog box of the node marker and type "label=out" in the parameter field). Now this node will be always be referred to as "out" and the voltage on that node as "v(out)". Editing the label parameter of a node marker is identical to editing it on a component--see "Editing Component Parameters" for information on changing the "label=" parameter.

Only node markers that are in the same subcircuit or main circuit will be connected together. Two node markers with the same name but in different subcircuits will not be connected together. The only connections allowed between different subcircuits or from the main circuit to a subcircuit are through terminals (see below).

Ground

Ground is a special type of node marker. Any node that a ground component is connected to will be shorted to the global node ground. Ground node is always given the number "0" by TRAVIS.

Terminals

Terminals are used in subcircuits to indicate which nodes connect to the terminals of the subcircuit body. Like components, terminals are automatically named as they are placed. You can change this name to any name you want. This name corresponds to the terminal names on the subcircuit body (see "Editing the Subcircuit Body"). You may have more than one terminal with the same name in the subcircuit; in this case both

terminals would be connected to the corresponding subcircuit body terminal (and, therefore, those two nodes would be shorted together). You may not, however, have two subcircuit body terminals with the same name (see "Editing the Subcircuit Body"). If a terminal is given a label that does not correspond to a terminal label on the subcircuit body, that node will not be connected to anything (the program will just ignore this terminal). The terminal names can be changed in the same way as node markers--just change the "label=" parameter in the Edit Component dialog box of the terminal. Just remember that you must also change the corresponding terminal name on the subcircuit body in order for the circuit terminal to remain connected to the body terminal.

Ports

Ports are used in port analyses to define the ports of a circuit at which the s-parameters will be calculated. These objects are used by TRAVIS only in a port analysis--they will be ignored in other analyses. See "Port Analysis" and "Port Analysis Results" for more information on how these are used in a port analysis.

3.7 EDITING COMPONENT PARAMETERS

Once a component is placed on the circuit, you will probably want to edit its parameters (e.g. resistor value, node marker name, etc.) To do this you must be in "Edit circuit" mode. Click the right mouse button until the status bar says "Edit circuit".

Now move the mouse pointer over the component you want to edit and push down the left mouse button. If the component is a primitive component, the Edit Component dialog box pops up when you release the mouse button (shown in Fig. 3.7.1). If the component is a subcircuit component, a floating menu will pop up with two selections: "Edit Component" and "Edit Subcircuit". To pop up the Edit Component dialog box for the subcircuit, select "Edit Component" and release the mouse button.

Resolved Information Field. The top text section of the Edit Component dialog box (the Resolved Information field) shows the node numbers of the component, the primitive name or subcircuit name, and the resolved parameters and variables used in the Parameter field. This information is valid only after a simulation is run--this is when TRAVIS evaluates parameter values and node numbers.

Parameter Field. The bottom section (the Parameter field) contains the parameter assignments that determine the parameter values the component will have when the next analysis is run. The Parameter field also contains any other information that you may wish to type, such as comments or personalized variables. For detailed information on allowed syntax, see "Parameter Assignment Syntax."

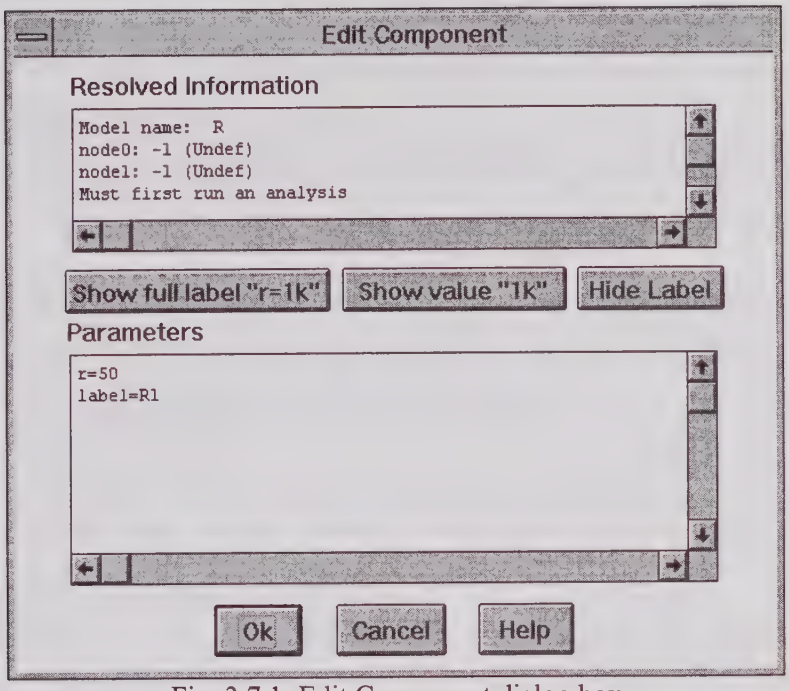
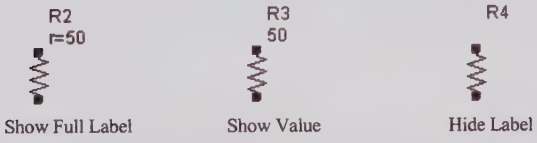


Fig. 3.7.1: Edit Component dialog box

Resolved Information Field. The top text section of the Edit Component dialog box (the Resolved Information field) shows the node numbers of the component, the primitive name or subcircuit name, and the resolved parameters and variables used in the Parameter field. This information is valid only after a simulation is run--this is when TRAVIS evaluates parameter values and node numbers.

Parameter Field. The bottom section (the Parameter field) contains the parameter assignments that determine the parameter values the component will have when the next analysis is run. The Parameter field also contains any other information that you may wish to type, such as comments or personalized variables. For detailed information on allowed syntax, see "Parameter Assignment Syntax."



Show Full Label. To make component parameters visible on the circuit schematic, click on the parameter assignment within the Component Parameter field. Then click on the "Show Full Label" button. The full label (e.g., "r=1k" or "td=50n") will appear next to the component on the schematic.

Show Value. Use this button the same way as the "Show Full Label" button above, except only the value of the parameter will appear (e.g., "1k" or "50n").

Hide Label. Use this button the same way as the "Show Full Label" button above, except clicking this button causes the parameter to disappear from the schematic.

OK. This button saves the current settings and closes the dialog box.

Cancel. This button closes the dialog box and restores all parameters to the way they were before you opened the dialog box (except for changes in label visibility).

For information about specific components, see "Primitive Components".

Parameter Assignment Syntax

The Parameter field is a free-style text editing window. To change component parameters, just edit the information appearing in this area. You can type anything you want into this area including comments--TRAVIS will ignore anything it does not recognize. You must enter one parameter per line. Also, if the same parameter appears on the left side of an equal sign more than once, TRAVIS will use only the first one in the list.

The general syntax of a parameter assignment is as follows:

<parameter name>=<expression>

The "=" sign may have spaces around it, but TRAVIS will remove these spaces after you close the Edit Component dialog box. No spaces are allowed within the <parameter name> or the <expression>--TRAVIS assumes that a space marks the end (or beginning) of names, constants, and expressions. For example, if you enter "stub length=100p", this is the same as entering "length=100p" because the program will not consider "stub" to be part of the parameter name. You need to type "stub-length" or "stub_length" to combine the two words.

<expression> can be any allowable mathematical expression containing multiple constants and variables.

For information on the allowable syntax of <expression>, see "Mathematical Expressions." For information on available parameter names of primitive components, see "Primitive Components." For information on variables and how they are defined, see "Variables."

Comments and Commenting Out Parameters

You may comment out a line in the Component Parameter field (the lower section of the Edit Component dialog box) by inserting a semicolon on that line. Everything after the semicolon on that line is ignored. This is handy when, for example, you have multiple "tran=" definitions in a voltage source and you want to just comment out the ones you aren't using at the moment. Additionally, any expression that TRAVIS does not recognize is ignored. For example, if you type a comment like "place near capacitor" in the

Parameter field of a component, it will be ignored (because TRAVIS doesn't know what it means) and so you don't need to use a semicolon.

Moving Component Labels

To move the labels around, close the Edit Component dialog box and just click the left mouse button on the labels themselves. The labels will become attached to the mouse pointer. Move the labels to the desired location. Click the left mouse button to place the labels.

Note that you can also make comments visible. For example, if you typed "place near capacitor" in the parameter assignment field as a circuit layout note and you want it displayed on the circuit schematic, just put the cursor on the line that contains that comment and click the "Show Full Label" button. (If you change this comment in the future, the label disappears and you have to click the "Show Full Label" button again to make it reappear.)

3.8 MATHEMATICAL EXPRESSIONS

In parameter and variable assignments, TRAVIS is able to interpret simple mathematical expressions such as:

2+25*5^2

There are five operators that can be used:

x+y	x plus y
x-y	x minus y
x*y	x multiplied by y
x/y	x divided by y
x^y	x raised to the power of y

Expressions are evaluated from left to right. Power operators are evaluated first, followed by multiply and divide, followed by plus and minus. This works the same as if you typed the expression on a scientific entry format calculator.

Parentheses may be used to force the expression to be evaluated in a certain way:

(2+25)*5

In this example, 2+25 is evaluated first, and the result of that is then multiplied by 5. Parentheses may be nested up to 20 deep.

Expressions may also contain variables, as in:

(2+x)*5

In general, no spaces are allowed within the expression. TRAVIS treats a space as the end of the expression.

See "Variables," "Editing Component Parameters," and "Subcircuit Parameters" for more information on when and where variables and expressions may be used.

3.9 VARIABLES

All variables arise from parameter assignments or expressions you type into the parameter fields of Edit Component dialog boxes (or the Edit Main Circuit Parameter dialog box).

All variables in TRAVIS are treated locally--that is, a variable defined in one subcircuit instance will not be visible in any other subcircuit instance. Variable assignments made in the parameter field of a primitive component are local to just that parameter field. Variable assignments made in the parameter field of a subcircuit component are not only available in that subcircuit component's parameter field, they are also passed by value down to the subcircuit within that subcircuit component--i.e., that variable is visible to all components in the underlying subcircuit. For example, suppose the variable assignment "Z=55" is typed into the parameter field of a subcircuit called "My_Circuit". "My_Circuit" might contain, among other things, a resistor "R1". If you type "r=Z" in the parameter field of "R1", the resistance of that resistor will be 55 ohms when an analysis is run. When a variable is passed in this way, we call it a subcircuit parameter. (See "Subcircuit Parameters" for more information.)

All error checking is done at analysis time. If TRAVIS can't resolve a variable (e.g., because it wasn't defined or there was a syntax error), it is flagged in the analysis status dialog box and in the resolved information field of the component containing the error. The resolved information field will also contain the final values of all that component's variables and parameters if you want to double-check them (you may have to scroll down to see them).

Mathematical expressions are allowed in variable assignments. So, in our example above, we could have entered "Z=55*2" and the resistor would have been 110 ohms instead. To get more fancy, we could have also entered "Z=55*n" if "n" were defined in the parameter field of the subcircuit or main circuit that contains "My_Circuit". (See "Mathematical Expressions" for information on allowable syntax for mathematical expressions.)

An example of where variables are useful is in the design of a generalized circuit whose parameters need to be changed often, but the structure of the circuit does not. For example, you might create a subcircuit that models the input characteristics of an IC pin. If you defined all component values within the subcircuit as a multiple of some variable (e.g., "c=1p*fanout" for all capacitors), then all you need to do to change the circuit loading would be to change the variable "fanout" in the subcircuit component's parameter field ("fanout=1" or "fanout=2").

3.10 PRIMITIVE COMPONENTS

The TRAVIS primitive components are listed below. Refer to the specific component for detailed parameter syntax.

Resistor
Capacitor
Inductor
Voltage source
Current source
Diode (professional version only)
Transmission line
Dependent voltage source
Dependent current source
Volt meter
Current meter

3.10.1 Resistor

Parameter	Syntax
resistance	r=<resistance>

Resistance specifies the resistance of the resistor in ohms. It must be greater than zero and causes an error if not specified.

Example:

r=1.5k sets the resistance to 1.5 kilo-ohms.

3.10.2 Capacitor

Parameter	Syntax
capacitance	c=<capacitance>

Capacitance specifies the capacitance of the capacitor in Farads. It must be greater than zero and causes an error if not specified.

Example:

c=10u sets the capacitance to 10 micro-Farads.

3.10.3 Inductor

Parameter	Syntax
inductance	l=<inductance>

Inductance specifies the inductance of the inductor in Henrys. It must be greater than zero and causes an error if not specified.

Example:

l=10.5n sets the inductance to 10.5 nano-Henrys.

3.10.4 Voltage source

Parameter	Syntax
dc voltage	dc=<dc voltage>
ac voltage magnitude	acm=<ac voltage magnitude>
ac voltage phase	acp=<ac voltage phase>
transient pulse voltage	tran=pulse(<iv>, <pv>, <td>, <tr>, <tf>, <pd>, <T>) iv = initial value pv = pulsed value td = delay time tr = rise time tf = fall time pd = pulse duration T = period

transient sinusoid	tran=sin(<os>, <amp>, <freq>, <delay>, <df>) os = offset amp = amplitude of sine wave freq = frequency of sine wave delay = delay to start of sine wave df = damping factor (When df < 0, the sine wave decays, when df > 0, the sine wave builds in amplitude.)

dc voltage specifies the dc voltage component of the voltage source in volts. It defaults to zero if not specified and is always used in ac and dc analysis. If the transient voltage is not specified, then it is also used in the transient analysis.

ac voltage magnitude specifies the magnitude of the ac component of the voltage source in volts. It is used in ac analysis and port analysis only. Strictly speaking, the magnitude is the peak voltage. The ac voltage magnitude defaults to zero if not specified.

ac voltage phase specifies the phase of the ac component of the voltage source in degrees. It is used in ac analysis and port analysis only, and it defaults to zero if not specified.

transient voltage specifies the pulse/wave shape of the voltage source. It is used in transient analysis only, and it defaults to zero if not specified. The figures below show graphically the meaning of the transient waveform parameters. Fig. 3.10.1 is for the "pulse" function and Fig. 3.10.2 is for the "sin" function. All of the time parameters must be greater than zero. Any of the parameters may be mathematical expressions.

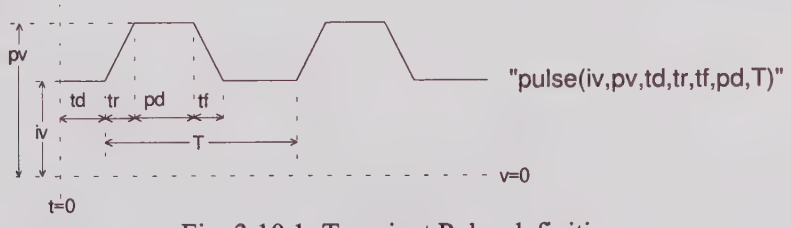


Fig. 3.10.1: Transient Pulse definition

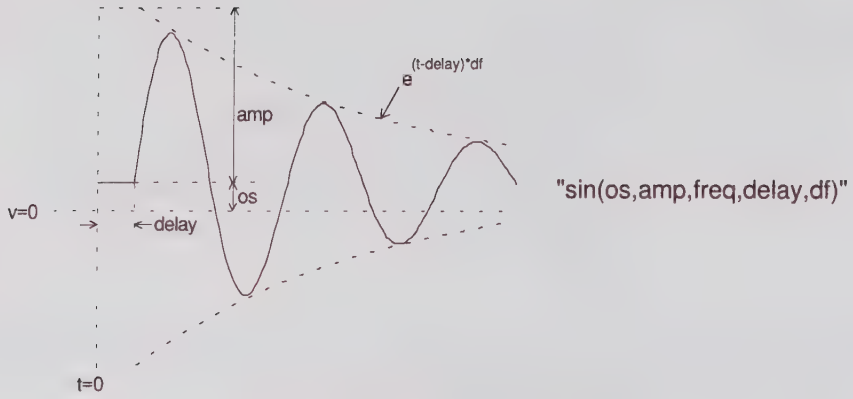


Fig. 3.10.2: Transient Sin definition

Examples:

dc=-5 sets the dc voltage to -5 volts.

acm=1 sets the ac magnitude to 1 volt.

acp=90 sets the phase to 90 degrees.

tran=pulse(0,1,1n,10n, 20n, 100n, 200n) specifies a transient pulse with an initial value of 0V and a pulsed value of 1V. The pulse begins at time point 1n. It has a rise time of 10n, fall time of 20n, pulse duration of 100n, and a period of 200n.

3.10.5 Current Source

Parameter	Syntax
dc current	dc=<dc current>
ac current magnitude	acm=<ac current magnitude>
ac current phase	acp=<ac current phase>
transient pulse current	tran=pulse(<iv>, <pv>, <td>, <tr>, <tf>, <pd>, <T>) iv = initial value pv = pulsed value td = delay time tr = rise time tf = fall time pd = pulse duration T = period

transient sinusoid	tran=sin(<os>, <amp>, <freq>, <delay>, <df>) os = offset amp = amplitude of sine wave freq = frequency of sine wave delay = delay to start of sine wave df = damping factor (When df < 0, the sine wave decays; when df > 0, the sine wave builds in amplitude.)

dc current specifies the dc current component of the current source in amps. It defaults to zero if not specified and is always used in dc and ac analysis. If the transient voltage is not specified, then it is also used in the transient analysis.

ac current magnitude specifies the magnitude of the ac component of the current source in amps. It is used in ac analysis and port analysis only. Strictly speaking, the magnitude is the peak current. The ac current magnitude defaults to zero if not specified.

ac current phase specifies the phase of the ac component of the current source in degrees. It is used in ac analysis and port analysis only and it defaults to zero if not specified.

transient current specifies the pulse/wave shape of the current source. It is used in transient analysis only, and it defaults to zero if not specified. See "Voltage Source" for a diagram of what the transient parameters represent. All of the time parameters must be greater than zero. Any of the parameters may be mathematical expressions.

Examples:

dc=-5m sets the dc current to -5 milliamps.

acm=1 sets the ac magnitude to 1 amp.

acp=90 sets the phase to 90 degrees.

tran=pulse(0,10u,1n,10n, 20n, 100n, 200n) specifies a transient pulse with an initial value of 0A and a pulsed value of 10uA. The pulse begins at time point 1n. It has a rise time of 10n, fall time of 20n, pulse duration of 100n, and a period of 200n.

3.10.6 Diode (professional version only)

Parameter	Syntax
saturation current	is=<saturation current>
emission coefficient	n=<coefficient>
series resistance	rs=<series resistance>
reverse breakdown voltage	BV=<breakdown voltage>
reverse breakdown current	IBV=<breakdown current>
temperature	T=<temperature>

Saturation current specifies the saturation current of the diode in amps. It must be greater than zero and causes an error if not specified.

Series resistance specifies the parasitic series resistance of the diode in ohms. It must be greater than or equal to one nano-ohm. If you specify between zero and one nano-ohm, then the simulator will set it to one nano-ohm; if you specify less than zeri, then an error is generated. The default value is one nano-ohm.

Emission coefficient specifies the emission coefficient of the diode. This is a unitless parameter. It must be greater than or equal to one. The default value is one.

Reverse breakdown voltage specifies the reverse breakdown voltage of the diode in volts. It must be greater than zero. The default value is 100V.

Reverse breakdown current specifies the reverse current that flows at the reverse breakdown voltage in amps. It must be greater than zero. The default value is 1mA.

Temperature specifies the junction temperature of the diode in degrees Celsius. It must be greater than -273ºC. The default value is 27ºC.

All diodes have a shunt conductance of 1e-12 Siemens that can not be changed by the user. This is included as part of the model to allow the simulator to converge on a solution and will not significantly affect analysis results.

Examples:

is=1e-16 sets the diode saturation current to 1e-16 amps.

rs=15 sets the parasitic series resistance of the diode to 15 ohms.

n=2 sets the emission coefficient to 2.

BV=10 sets the reverse breakdown voltage to 10 volts.

IBV=10mA sets the reverse breakdown current to 10mA.

T=125 sets the junction temperature to 125ºC.

3.10.7 Transmission Line

Parameter	Syntax
propagation length	td=<length in time>
characteristic Impedance	z0=<impedance in ohms>
type of transmission line	type=<type>
max animation voltage	vmax=<volts>
min animation voltage	vmin=<volts>
max animation current	imax=<amps>
min animation current	imin=<amps>
max animation phase	pmax=<degrees>
min animation phase	pmin=<degrees>

The transmission line symbol is marked with a small axis (a pair of perpendicular arrows). This is meant to indicate the origin of the transmission line for plotting purposes--i.e., this corner represents zero time on the transmission line's horizontal axis. This marker is handy if you have mirrored or rotated the transmission line. (See "Animating Transient Animation Plots" and "Animating AC Animation Plots" for more information.)

Propagation length specifies the length of the transmission line in seconds (time). It must be greater than zero. An error is generated if this parameter is not specified.

Characteristic Impedance specifies the characteristic impedance of the transmission line in ohms. It must be greater than zero. An error is generated if this parameter is not specified.

Type of transmission line indicates the geometry type of the transmission line. The allowed types are "microstrip", "stripline", "coax", and "twinlead". This will not affect any analysis, but it does affect the Transmission Line Designer tool.

Max and min animation voltage, current, and phase are used to specify the limits of the animation plot for that transmission line (if there is an animation plot). pmin and pmax are only used in ac analysis. Quite often it is easiest to let TRAVIS automatically set these values for you--for information on how to do this, see the description of the "Setup Transient Animation Plot" and "Setup AC Animation Plot" menu items under the "See Results" menu. For more information on transmission line analysis and animation, see "Animating Transient Animation Plots" and "Animating AC Animation Plots".

3.10.8 Dependent Voltage Source

Parameter	Syntax
voltage dependency function	v=depfunc(<driver source>, <multiplication factor>)

Voltage dependency function specifies which circuit voltage or current this source is dependent on, and how it is dependent on it. The <driver source> is simply the label of the voltage or current meter elsewhere in the circuit that the dependent source depends on. That voltage or current will be multiplied by the <multiplication factor> to obtain the actual voltage of the dependent source during analysis. TRAVIS automatically senses whether the <driver source> is a current or voltage. If the driver source is a voltage, then a voltage amplifier is formed. If the driver source is a current, then a trans-resistance is formed. The multiplication factor may be a mathematical expression.

Examples:

v=depfunc(VM1,5) (voltage dependent voltage source) says that this voltage source is equal to 5 times the voltage across the voltage meter VM1. If VM1 is not present somewhere on the local circuit (or subcircuit), an error will be generated by TRAVIS.

v=depfunc(IM2,4.5k) (current dependent voltage source--trans-resistance) says that this voltage source is equal to 4500 times the current through the current meter IM2. If IM2 is not present somewhere on the local circuit (or subcircuit), an error will be generated by TRAVIS. You can use the name of a voltage source or other dependent voltage source as well.

3.10.9 Dependent Current Source

Parameter	Syntax
current dependency function	i=depfunc(<driver source>, <multiplication factor>)

Current dependency function specifies which circuit voltage or current this source is dependent on, and how it is dependent on it. The <driver source> is simply the label of the voltage or current meter elsewhere in the circuit that the dependent source depends on. That voltage or current will be multiplied by the <multiplication factor> to obtain the actual current of the dependent source during analysis. TRAVIS automatically

senses whether the <driver source> is a current or voltage. If the driver source is a voltage, then a trans-conductance is formed. If the driver source is a current, then a current amplifier is formed. The multiplication factor may be a mathematical expression.

Examples:

i=depfunc(VM1,5m) (voltage dependent current source--trans-conductance) says that this current source is equal to 5e-3 times the voltage across the voltage meter VM1. If VM1 is not present somewhere on the local circuit (or subcircuit), an error will be generated by TRAVIS.

i=depfunc(IM2,4.5) (current dependent current source) says that this current source is equal to 4.5 times the current through the current meter IM2. If IM2 is not present somewhere on the local circuit (or subcircuit), an error will be generated by TRAVIS. You can use the name of a voltage source or dependent voltage source as well.

3.10.10 Volt Meter

There are no parameters needed for a volt meter unless it is being referred to by a dependent source, and then only its label needs to be present (all components must have a label anyway). The volt meter itself has no effect on the circuit.

A volt meter has two purposes: 1) Act as a voltage sensor for a dependent voltage source or a dependent current source, and 2) Act as a component that can be clicked on to obtain the difference between two node voltages (see the "Results" menu).

3.10.11 Current Meter

There are no parameters needed for a current meter unless it is being referred to by a dependent source, and then only its label needs to be present (all components must have a label anyway). The current meter itself causes a short circuit between it's terminals.

A current meter has two purposes: 1) Act as a current sensor for a dependent voltage source or a dependent current source, and 2) Act as a component that can be clicked on to obtain the current through a branch of the circuit (see the "Results" menu).

3.11 EDITING THE CIRCUIT

There are two types of circuits in TRAVIS: 1) the main circuit, and 2) subcircuits. For the purposes of editing the actual circuit, the two are absolutely identical.

Once a circuit is created, you may want to change the way it looks by performing cut, paste, or move operations on it. In order to do this, you will have to select the part of the circuit you want to edit.

To select a portion of the circuit for editing:

1. You must first have the circuit editor in "Edit circuit" mode. This is done by clicking the right mouse button until "Edit Circuit" appears in the status text in the upper left corner of the circuit window.
2. Push down and hold the left mouse button at a corner of the area you want to select.
3. While holding down the left mouse button, drag the mouse. A selection rectangle is drawn to indicate the area that will be selected when you release the left mouse button. Drag the mouse until you've outlined the portion of the circuit you want to select.
4. Release the left mouse button. Everything that was in the selection rectangle is selected (selected components are highlighted). Only components that were entirely within the selection rectangle will be selected.

You can now copy, cut, or move the selected circuitry. See the descriptions of the "Copy", "Cut", "Paste", and "Move" menu items under the "Edit" menu for a detailed explanation of each of these editing operations. To unselect the selected circuitry, click the right mouse button.

3.12 SUBCIRCUITS

Subcircuits provide the ability to neatly organize the project in a hierarchical structure. In addition, sections of circuitry that are repetitive can easily be duplicated by using subcircuits. This also saves memory because the program stores only one copy of each subcircuit.

All subcircuits consist of: 1) the subcircuit itself, 2) a body (also called the subcircuit component) to represent the subcircuit when it is placed in the circuit hierarchy, and 3) the subcircuit parameters. Information on all of these topics is provided in the following sections.

3.12.1 Creating and Editing a Subcircuit

To create a subcircuit, you simply draw out and edit the main circuit schematic in the usual manner. The difference is now you will add terminals to your main circuit, draw a shape to represent your circuit (subcircuit body), and then select "Save Subcircuit As" from the "File" menu. TRAVIS just saves the current project as a subcircuit. This means that a file is created with the ".cir" extension (called a circuit file) that contains just the information about the circuit--i.e., the subcircuit body and circuitry itself--no analysis-related information is in this file. In addition, this file may later be placed as a subcircuit in another circuit (as described under "Placing Subcircuit Components").

To edit an exisiting subcircuit or its body, select the "Open Subcircuit" menu item from the "File" menu.

3.12.2 Subcircuit Parameters

Subcircuit parameters work the same way as primitive component parameters--the only difference is that they are user specified. Subcircuit parameter is simply another name for a variable that is used within the subcircuit.

To edit the value of the subcircuit parameters, simply click on the subcircuit component and edit the parameters the same way you would for a primitive component--i.e., by typing in the parameter field (see "Editing Component Parameters").

You create and define a subcircuit parameter simply by using it in the subcircuit itself (e.g., in the parameter field of a resistor, type "r=Rin", where Rin is a parameter you want for your subcircuit). Usually, you'll want to define a default value for the parameter. This is done by selecting the "Main Circuit Parameters" item from the "Edit" menu when the subcircuit is being edited as the main circuit of the project.

As an example, you might have an amplifier subcircuit whose input resistance you want to specify with a subcircuit parameter called "inputZ" so that it can be conveniently changed. Assuming that the input resistance is set by a single resistor in the subcircuit, first open the Edit Component dialog box for that resistor. Then edit the resistance of that resistor by entering "r=inputZ". Next, you will want to specify a default value for "inputZ". To do this, go to the "Edit" menu (in the main circuit window) and select the "Main Circuit Parameters" item. In the parameter field, type in "inputZ=1M"--assuming you want the default value to be 1Mohm. Now you can save the circuit as a subcircuit (see "Save Subcircuit"). Now when you place the amplifier in another circuit hierarchy and run an analysis, the value of the input resistor will be set equal to the value of "inputZ", which is in turn set equal to 1M by default. You can also change "inputZ" to any other value (by changing its assignment in the parameter field of the amplifier's subcircuit component); when the analysis is rerun, the new value of inputZ will be assigned to the value of the resistor.

3.12.3 Terminals

Terminals are used in subcircuits to indicate which nodes in the subcircuit connect to the terminals of the subcircuit body. Like components, terminals are automatically named as they are placed. You can change this name to anything you want. This name corresponds to the terminal names on the subcircuit body (see "Editing the Subcircuit Body"). You may have more than one terminal with the same name in the subcircuit, in which case the two terminals are both connected to the corresponding subcircuit body terminal, and, therefore, those two nodes will be shorted together. You cannot, however, have two subcircuit body terminals with the same name (see "Editing Subcircuits"). If a terminal is given a label that does not have a corresponding terminal name on the subcircuit body, then that node will not be connected to anything (the program will just ignore this terminal). The terminal names are edited the same way as node markers--just change the "label=" parameter assignment. Remember that you must also change the corresponding terminal name on the subcircuit body in order for the terminal to remain connected to that body terminal.

3.12.4 Editing the Subcircuit Body

The subcircuit body editor is opened by selecting "Body" from the "Edit" menu of the main circuit window. The subcircuit body may be constructed from lines, rectangles, arcs, and terminals. Terminals correspond to the terminals in the subcircuit. The editor automatically numbers the terminals as they are placed; however, they may be renamed. The terminal names must correspond to the terminal names in the subcircuit schematic. To change a terminal name, click the left mouse button on the terminal itself. A small edit box pops up to allow you to change the name.

To move a terminal name around, click the left mouse button on the name itself (first make sure the status text displays "Edit body"--this is done by clicking the right mouse button). The terminal name attaches itself to the mouse pointer. When you have moved to the desired location, click the left mouse button again.

The subcircuit body cannot be drawn infinitely large. If you scroll far enough, you will encounter dashed lines that represent the boundaries of the drawing space.

When you are done drawing the subcircuit body, just close the subcircuit body editor. The body drawing is automatically saved along with the subcircuit when you select the "Save Subcircuit" item from the "File" menu.

3.12.5 Edit Subcircuit Body Menus

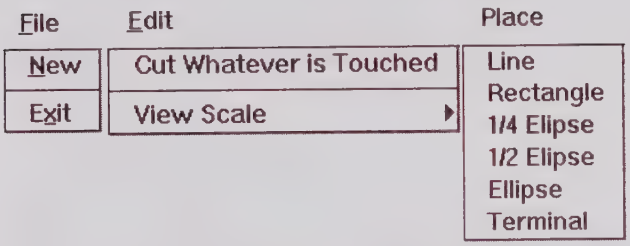


Fig. 3.12.1: Subcircuit Body window menus

The menu items in the subcircuit body window are described below.

File menu

New
Selecting this menu item deletes the subcircuit body.

Exit
Selecting this menu item closes the subcircuit body window. Any changes made to the subcircuit body are saved.

Edit menu

Cut Whatever Is Touched
After selecting this menu item, you can delete any piece of the body by clicking the left mouse button on that piece.

View Scale
Selecting this menu item causes a submenu to pop up with scaling selections ranging from 50% to 400%. 100% is so-called nominal scaling and is what the editor starts up in. Selecting less than 100% will cause the drawing to appear smaller (zoom out), resulting in more of the drawing being visible but with less detail. Selecting more than 100% will cause the drawing to appear larger (zoom in), resulting in less of the drawing being visible but with more detail.

Place menu

Line
After selecting this menu item, you are prompted to click the starting location of the line. Then you are prompted to click the ending location of the line. A single line will be drawn between the two locations. Click the right mouse button to leave this mode.

Rectangle
After selecting this menu item, you are prompted to click the starting location of the rectangle. Then you are prompted to click the ending location of the rectangle. A single rectangle will be drawn between the two locations. Click the right mouse button to leave this mode.

1/4 Ellipse
After selecting this menu item, you are prompted to click the starting location of the 1/4 ellipse. Then you are prompted to click the ending location of the 1/4 ellipse. Before clicking the ending location, you can change the orientation of the 1/4 ellipse by hitting the space bar. A single 1/4 ellipse will be drawn between the two locations. Click the right mouse button to leave this mode.

1/2 Ellipse
After selecting this menu item, you are prompted to click the starting location of the 1/2 ellipse. Then you are prompted to click the ending location of the 1/2 ellipse. Before clicking the ending location, you can change the orientation of the 1/2 ellipse by hitting the space bar. A single 1/2 ellipse will be drawn between the two locations. Click the right mouse button to leave this mode.

Ellipse
After selecting this menu item, you are prompted to click the starting location of the ellipse. Then you are prompted to click the ending location of the ellipse. Before clicking the ending location, you can change the orientation of the ellipse by hitting the space bar. A single ellipse will be drawn between the two locations. Click the right mouse button to leave this mode.

Terminal
After selecting this menu item, you simply click the mouse button in the locations you want terminals. Click the right mouse button to leave this mode.

3.12.6 Placing Subcircuit Components

Although placing a subcircuit component is the same as placing a primitive component, there are a few differences between subcircuit components and primitive components that should be mentioned. For one thing, a subcircuit component represents a subcircuit made up of primitive components and perhaps other subcircuit components. Also, subcircuit components are retrieved from disk, whereas primitive components are hardwired into TRAVIS. Finally, you can edit the appearance of a subcircuit component as well as its contents (see "Creating and Editing a Subcircuit").

There are two ways to add a subcircuit component to the current circuit: 1) get a new subcircuit from disk, or 2) place a subcircuit that is already loaded.

Getting a new subcircuit from disk

1. Pull down the "Place" menu and select "New Subcct Component from Disk". An Open File dialog box pops up listing all files with file extensions ".cir" (see the de-

scription of the "Open Subcircuit" menu item for information on the controls in this dialog box).

2. Select the file you want and click "OK".
3. Now the subcircuit component (body) is attached to the mouse pointer. You can rotate, mirror, and place the subcircuit the same as a primitive component (see "Placing Components").

Your project maintains its own private copy of this subcircuit within the Subcircuit Library, which will be saved to disk when you select "Save Project".

Placing a previously loaded subcircuit ("Quickref" menu)
Whenever you get a new subcircuit from disk by selecting the "New Subcct Component from Disk" menu item, this subcircuit is put into the project's Subcircuit Library as described above. However, if you want to place more copies of this subcircuit component in your project later on, you don't have to load it from disk again, you can instead:

1. Select the "Subcircuit Library" menu item from the "Edit" menu. The Subcircuit Library dialog box pops up.
2. Select the name of the subcircuit you want from the left-hand list box by clicking on it with the left mouse button.
3. Then click the "Place Selection" button in the lower portion of the dialog box. The dialog box closes and the subcircuit will be attached to the mouse pointer so you can place it in the circuit.

Adding items to the "Quickref" menu
If you will be using this subcircuit often in this project, you will want to put it in the subcircuit component "Quickref" menu. To do this:

1. Select the "Subcircuit Library" menu item from the "Edit" menu. The Subcircuit Library dialog box pops up.
2. Select one of the subcircuits from the left-hand list box by clicking on it with the left mouse button.
3. Now click the "Add Selection to Quickref Menu" button. The subcircuit will now also appear in the right-hand list box. Close the dialog box by clicking "OK". The item has been added to the "Quickref" menu.
4. To place that component using the "Quickref" menu, pull down the "Place" menu and select the "Subcct Component Quickref" menu item, which expands into a submenu containing the subcircuits in the "Quickref" menu. Select one of the components. The subcircuit will be attached to the mouse pointer so you can place it in the circuit.

You can have up to 50 subcircuit components in the "Quickref" menu.

3.12.7 Editing a Subcircuit Already Placed in the Circuit

You may want to edit a subcircuit that has been created and loaded previously and that you have placed into the circuit you are currently editing. To edit the subcircuit, first get into "Edit circuit" mode by clicking the right mouse button until the status bar says "Edit circuit".

Then move the mouse pointer over the subcircuit component whose subcircuit you want to edit and push down the left mouse button. If the component is indeed a subcircuit component (i.e., not a primitive component), a floating menu will pop up with two selections: "Edit Component" and "Edit Subcircuit". Select "Edit Subcircuit" from the menu and release the mouse button.

Another circuit window will pop up with the subcircuit in it. You can now edit the subcircuit just as you would any other circuit.

Remember that the changes you make to the subcircuit apply to all instances of this subcircuit in your project. If you have opened multiple instances of the same subcircuit and you change the subcircuit in one, it will change in the others also (however, you need to select "Refresh" from the "Edit" menu to be sure these changes are visible).

You may not edit the body of this subcircuit or any subcircuit that is in the Subcircuit Library. If you want to do this, you must close the current project and open the subcircuit as a subcircuit (see "Open Subcircuit"). Remember that since the subcircuit library is private to this project, any changes you make to a subcircuit in the current project are not reflected in the *.cir file on disk. For this to occur, you must select the "Subcircuit Library" item from the "Edit" menu and save the subcircuit from the library to disk. Likewise, any changes made to circuit (*.cir) files on disk will have no effect on corresponding subcircuits in a subcircuit library. The subcircuits must be reloaded from disk (as described under "Getting a New Subcircuit from Disk").

3.12.8 Subcircuit Library

The subcircuit library is a repository of subcircuits that have been loaded into the project. It is local to the project and, therefore, any changes you make to the subcircuits will be reflected only in the project and not in any *.cir files on disk.

In order to access the subcircuit library, you need to pull down the "Edit" menu and select the "Subcircuit Library" item. This pops up the Subcircuit Library dialog box.

Subcircuit Library Dialog Box

This dialog box contains two list boxes. The left-hand box contains a list of all subcircuits in the subcircuit library (see "Getting a New Subcircuit from Disk" for information on how these subcircuits get here in the first place).

The right list box contains a list of subcircuit components that are in the "Quickref" menu. (see "Placing a Previously Loaded Subcircuit").

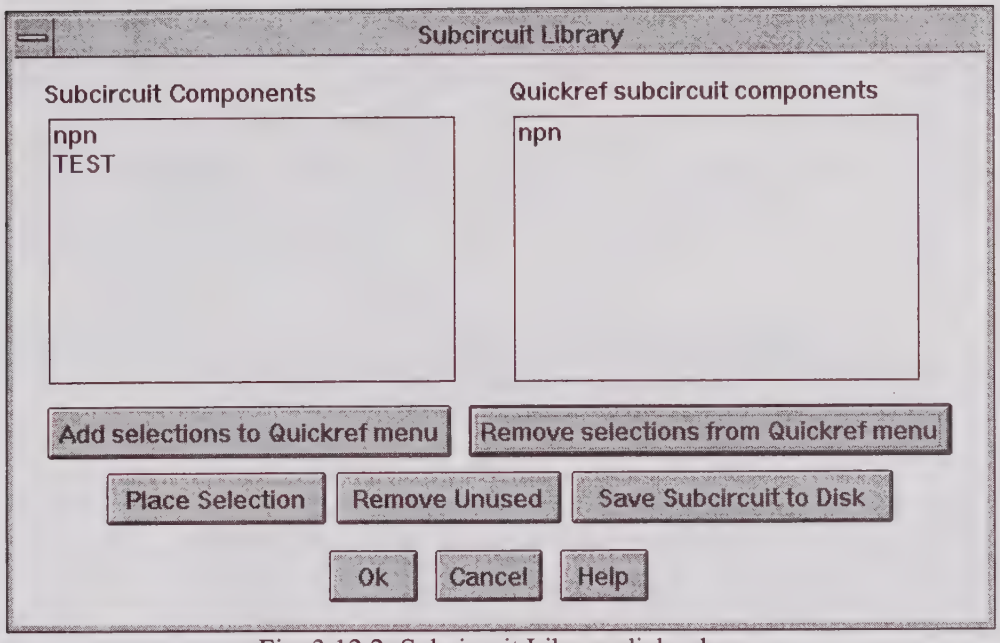


Fig. 3.12.2: Subcircuit Library dialog box

Buttons:

Add Selections to Quickref Menu: A subcircuit component selected in the left list box is placed in the "Quickref" menu and shows up in the Quickref Subcircuit Components list box.

Remove Selections from Quickref Menu: A subcircuit component selected in the right list box is removed from the "Quickref" menu.

Place Selection: The dialog box will close, and the subcircuit component selected in the left list box will be attached to the mouse pointer ready to be placed in the circuit (see "Adding Subcircuit Components"). All other changes to the library will be remembered (as if you clicked the "OK" button).

Remove Unused: The entire project is scanned. Any subcircuit not currently being used in the main circuit (or in a subcircuit that is in the main circuit hierarchy) is removed from the library. A dialog box pops up to inform you of how many subcircuits will be removed from the library. Click the "OK" button in that dialog box. The deletions made from the library don't actually occur until you click the "OK" button described below. If you hit the "Cancel" button, the deletions are not done.

Save Subcircuit to Disk: This button allows you to save a subcircuit from the Subcircuit Library to disk. Simply select one of the subcircuits (subcircuit components) in the left-hand list and then click this button. The standard Save Subcircuit dialog box pops up. This is handy if you have modified the subcircuit within the main circuit and you want to save it to be used in other main circuits.

Ok: This button saves the current state of the library and the "Quickref" menu and closes.

Cancel: This button undoes all changes or deletions to the library and "Quickref" menu and closes.

3.13 UNITS

TRAVIS interprets the following letters in mathematical expressions:

a = atto = 10e-18
f = femto = 10e-15
p = pico = 10e-12
n = nano = 10e-9
u = micro = 10e-6
m = milli = 10e-3
k = kilo = 10e+3
M = mega = 10e+6
meg = mega = 10e+6 (M and meg are equivalent)
G = giga = 10e+9
T = tera = 10e+12

You may also explicitly type in the "e" as in 5e+17 or 2.38e-21. Either an upper- or lower-case "e" is allowed here.

3.14 TRANSMISSION LINE DESIGNER

This is a simple graphical tool that helps provide convenient impedance calculation and synthesis of various types of transmission lines. It can be accessed either by directly selecting it from the "Tools" menu or by clicking on a transmission line while the status text displays "Click t-line for T-Line Designer." (See "Edit Transmission Line Info" item in the "Edit" menu). In the latter case, the impedance and type of that transmission line on your circuit is transferred to the transmission line designer and placed in the "Zo" box.

Four different types of transmission lines are supported:

1. Microstrip
2. Stripline
3. Coaxial
4. Twin Lead

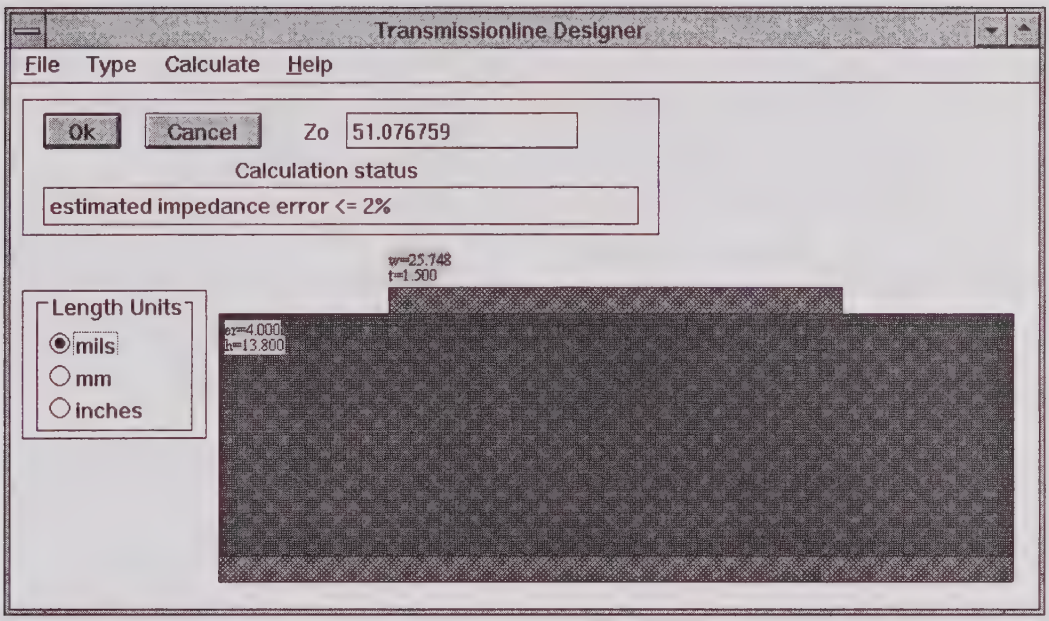


Fig. 3.14.1: Transmission Line Designer

On the left-hand side of the window is a list of buttons that allow you to change the displayed units. The three selections are mils (0.001 inch), inches, and millimeters. Just select the one you are most familiar with.

At the top of the window is the characteristic impedance (Zo) and a message display. Note that the Zo displayed here correctly corresponds to the cross section drawing only after performing a calculation by selecting one of the items from the "Calculate" menu (except for the "Plot" item). The message display might display a message such as the percentage error of the most recent calculation or whether the calculation was within the required range of the transmission line designer formulas. Similar messages are displayed under the trace track value of plot windows when you are trace tracking with the cursor keys.

Transmission line dimensions can be edited by clicking on the picture of the transmission line. A dialog box will pop up with the parameters that can be edited. (The Microstrip Parameter dialog box is shown in Fig. 3.14.2 as an example; the other types of transmission lines have similar dialog boxes.)

The check boxes on the left-hand side of the dialog box under the "visible" column are used to control whether the parameter values will be visible on the picture of the transmission line. When they are checked, the parameter will be visible.

Enter the values of the parameters in the appropriate spaces and click the "OK" button to accept the new values. Click the "Cancel" button to ignore any changes and close the dialog box.

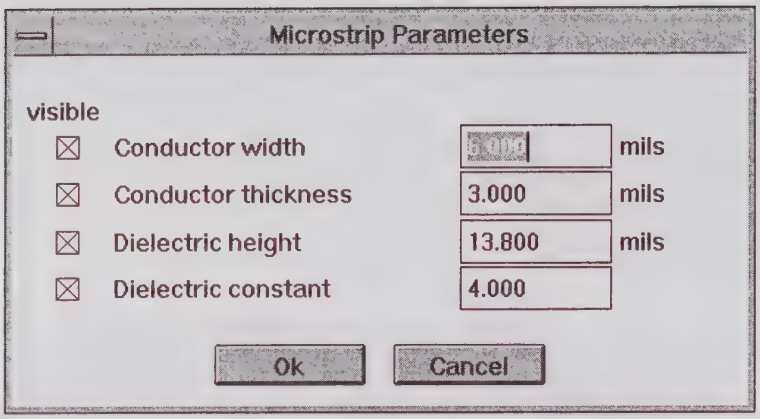


Fig. 3.14.2: Microstrip Parameter dialog box

3.14.1 Menus

The menus and menu items in the Transmission Line Designer are described below.

File menu

Exit
To exit the Transmission Line Designer, select the "Exit" menu item from the "File" menu.

Type menu

The Transmission Line Designer supports four types of dual-conductor (single-mode) transmission lines: Microstrip, Stripline, Coax, and Twin Lead. To edit the physical parameters, just click on the displayed picture of the transmission line cross section. Check boxes are provided to let you choose which parameters are displayed in the diagram--e.g., if you want only thickness to be displayed, then check only the box next to dielectric thickness.

The graphical depiction of all of the transmission lines is roughly to scale so you can get an expanded idea of what the cross section really looks like. Green represents the insulating material (the dielectric) and yellow-brown represents the conductor.

Microstrip
Microstrip is composed of a metal trace over a ground plane with an insulating material separating the two. Air is assumed to exist above the trace.

The microstrip physical parameters are:

Parameter	Parameter description	Allowed values
w	width of the conductor	0.05 <= w/h <= 50
t	thickness of the conductor	0 <= t/h <= 0.5
er	relative dielectric constant of the insulator	1 <= er <= 30
h	height (or thickness) of the insulator	0 <= t/h <= 0.5; 0.05 <= w/h <= 50

Stripline

Stripline is composed of a metal trace sandwiched between two insulating layers that are in turn sandwiched between metal ground planes.

The stripline physical parameters are:

Parameter	Parameter description	Allowed values
w	width of the conductor	0.05 <= w/2*h <= 50
t	thickness of the conductor	0 <= t/2*h <= 0.5
er	relative dielectric constant of the insulator	1 <= er <= 30
h	height (or thickness) of one insulator	0.05 <= w/2*h <= 50; 0 <= t/2*h <= 0.5

It is assumed that both layers of the insulator (one above and one below the trace) are made of the same material. The "h" (height) parameter is also assumed to be the same for the top and bottom layers of the insulator. Note that the total height of both insulators is 2*h.

Coax

Coax is composed of a cylindrical ground conductor surrounding the center conductor (usually carrying the signal). An insulator fills the space between the two.

The coax physical parameters are:

Parameter	Parameter description	Allowed values
outer	radius of the outer conductor	outer < 1000 * inner
inner	radius of the center conductor	outer < 1000 * inner
er	relative dielectric constant of the insulator	1 <= er <= 30

The outer radius is measured from the center of the coax to the inside surface of the shield (i.e., that surface facing the center conductor).

Twin Lead

Twin lead is simply two parallel round conductors.

The twin lead physical parameters are:

Parameter	Parameter description	Allowed values
radius	radius of the conductors	spacing < 1000 * radius
spacing	center-center spacing of the conductors	spacing < 1000 * radius
er	relative dielectric constant of the insulator	1 <= er <= 30

The conductors are both of the same radius.

Calculate menu

When any of these menu items are selected, that parameter is calculated using the other parameters. The estimated error of the Zo parameter (given the other parameters are correct) is also displayed.

Zo
The characteristic impedance is calculated and displayed using the displayed physical parameters of the transmission line.

w
The width of the conductor is calculated and displayed using the other physical parameters of the transmission line. The graphical drawing is also rescaled to indicate the change. This applies only to microstrip and stripline.

h
The height of the insulator is calculated and displayed using the other physical parameters of the transmission line. The graphical drawing is also rescaled to indicate the change. This applies only to microstrip and stripline.

t
The thickness of the conductor is calculated and displayed using the other physical parameters of the transmission line. The graphical drawing is also rescaled to indicate the change. This applies only to microstrip and stripline.

er
The dielectric constant of the insulator is calculated and displayed using the other physical parameters of the transmission line.

radius
The radius of the conductor is calculated and displayed using the other physical parameters of the transmission line. The graphical drawing is also rescaled to indicate the change. This applies only to twin lead.

spacing
The spacing of the two conductors is calculated and displayed using the other physical parameters of the transmission line. The graphical drawing is also rescaled to indicate the change. This applies only to twin lead.

inner radius
The radius of the center conductor is calculated and displayed using the other physical parameters of the transmission line. The graphical drawing is also rescaled to indicate the change. This applies only to coax.

outer radius
The radius of the outer conductor (shield) is calculated and displayed using the other physical parameters of the transmission line. The graphical drawing is also rescaled to indicate the change. This applies only to coax.

Plot
To make a plot of one transmission line parameter versus another (with the other parameters held constant), select the "Plot" menu item from the "Calculate" menu.

A dialog box pops up with three regions. The top two regions are Horizontal Axis and Vertical Axis. Click on the appropriate button in each region to select the transmission line parameters that will be along the vertical and horizontal axis. In the bottom region (Horizontal Axis Limits) you choose the range of the horizontal axis variable and the number of points to calculate within this range. The appropriate parameters are made available depending on the type of transmission line you have selected. The units are assumed to be those selected with the radio buttons in the main window of the Transmission Line Designer.

Clicking the "Run Plot" button causes a plot window to pop up with the information requested. Clicking the "OK" button simply saves the settings in the dialog box and closes it. Clicking "Cancel" closes the dialog box without remember the settings.

The plot window that pops up behaves just like any other TRAVIS plot window--there are just different units on the x- and y-axis. Plot windows originating from the Transmission Line Designer tool have a message displayed under the trace track value when you are trace tracking with the cursor keys. This message indicates the validity of the current trace point and might include messages such as the percentage error of the calculation for that point or whether the physical parameters were within the required range of the transmission line designer formulas for that data point. This message is useful in ascertaining the valid portions of the plot.

Chapter 4

Preparing for and Running the Analysis

Once you have entered a circuit (see "Entering and Editing a Circuit") you are ready to set up the analysis. There are four types of analysis in TRAVIS: 1) dc analysis, 2) ac analysis, 3) port analysis, and 4) transient analysis. Transmission line analysis (for animation) is done during ac and transient analysis.

4.1 DC ANALYSIS

dc analysis uses the "dc" parameter found in current sources and voltage sources to excite the circuit. The dc analysis is used when you just want to see the dc operating characteristics of the circuit. A dc analysis can also useful to make sure that you've properly drawn a circuit before starting a full-fledged ac or transient analysis. There is no setup to do a dc analysis.

To start a dc analysis:

1.) Select the "Run DC Analysis" item from the "Analysis" menu.

2.) The Analysis Status dialog box pops up to display the progress of the analysis. Click the "OK" button when the analysis is complete.

3.) The status text will display "Click wire/component for dc value."

Now you're ready to see the results of the analysis. For more information, see "DC Results".

4.2 AC ANALYSIS

Ac analysis uses the "acp" and "acm" parameters found in current sources and voltage sources to excite the circuit. For information about defining the ac analysis parameters (such as start and stop frequency and number of analysis points), see the description of

the "Setup AC Analysis" menu item under the "Analysis" menu. Ac transmission line analysis is also done during the ac analysis if you have it enabled.

To start the ac analysis (after setting it up):

1.) Select the "Run AC Analysis" item from the "Analysis" menu. (You could equivalently click the "OK-Run AC Analysis" button in the Setup AC Analysis dialog box.)

2.) The Analysis Status dialog box pops up to display the progress of the analysis. Click the "OK" button when the analysis is complete.

3.) The status text will display "Click wire/component for ac rectangular plot." You can view polar plots instead by selecting "AC Plot->Polar" from the "See Results" menu.

Now you're ready to see the results of the analysis. For more information, see "AC Results".

4.3 PORT ANALYSIS

Port analysis is a special form of ac analysis and so uses the settings found in the "Setup AC Analysis" dialog box. The biggest difference is that you add port components to your circuit to indicate the circuit ports (see "Placing Grounds, Node Markers, Terminals, and Ports"). One- and two-port analysis are supported.

TRAVIS will use only ports with the label "Port1" or "Port2". If two ports have the same name, then an analysis run time error is generated. Likewise, if no ports have either one of these names, an analysis run time error is generated. For more information on port components, see "Placing Grounds, Node Markers, Terminals, and Ports".

To start the port analysis (after setting it up):

1. Select the "Run Port Analysis" item from the "Analysis" menu. (You could equivalently click the "OK-Run Port Analysis" button in the Setup AC Analysis dialog box.)

2. The Analysis Status dialog box pops up to display the progress of the analysis. If you are doing a two-port analysis, notice that TRAVIS automatically runs two analyses--one for each port. Click the "OK" button when the analysis is complete.

3. The status text will display "Click wire/component for port polar plot." You can view rectangular plots instead by selecting "Port Plot->Rectangular" from the "See Results" menu.

Now you're ready to see the results of the analysis. For more information, see "Port Results".

4.4 TRANSIENT ANALYSIS

Transient analysis uses the "tran=" parameter found in current sources and voltage sources to excite the circuit. Transient transmission line analysis is done during a transient analysis if you enabled it in the Setup Transient Analysis dialog box along with all other transient analysis parameters.

To start the transient analysis (after setting it up):

1. Select the "Run Transient Analysis" item from the "Analysis" menu. (You could equivalently click the "OK-Run Transient Analysis" button in the Setup Transient Analysis dialog box.)
2. The Analysis Status dialog box pops up to display the progress of the analysis. Click the "OK" button when the analysis is complete.
3. The status text will display "Click wire/component for transient plot."

Now you're ready to see the results of the analysis. For more information, see "Transient Results".

4.5 ANALYSIS MENU

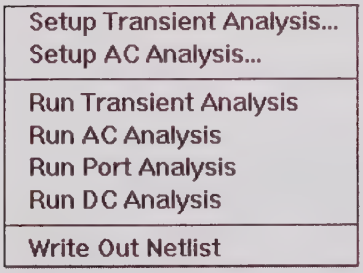


Fig. 4.5.1: Analysis menu

The analysis menu items (of the circuit window) are described below:

4.5.1 Setup AC Analysis

The Setup AC Analysis dialog box is used to define the parameters for the ac and port analysis. The contents of the Setup AC Analysis dialog box are detailed below. Changes made in this dialog box are applied to the next analysis.

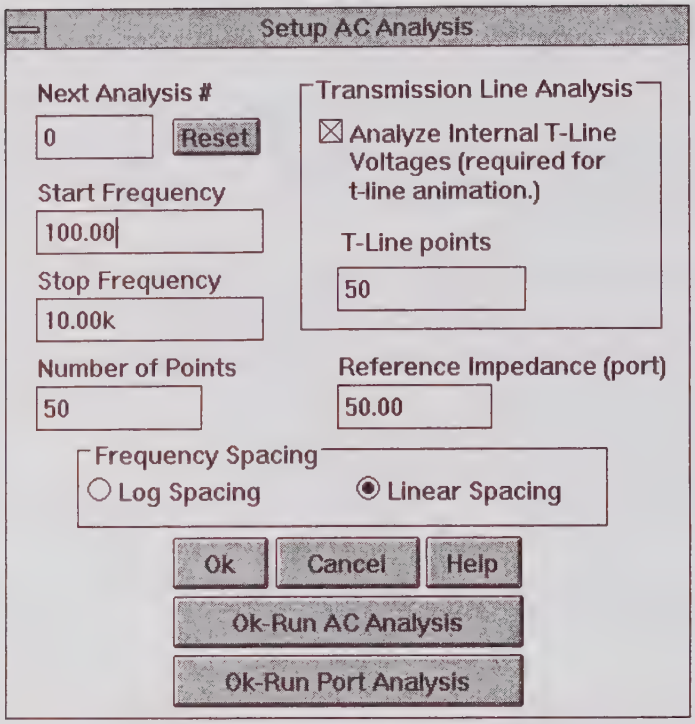


Fig. 4.5.2: Setup AC Analysis dialog box

Start frequency is the beginning frequency in the analysis.

Stop frequency is the end frequency in the analysis.

Number of points refers to the number of analysis points to be generated from start frequency to stop frequency (inclusive). You may not have more than 15,000 points.

Frequency spacing
Log spacing: This selection will cause the analysis points to be spaced in a logarithmic fashion--i.e., they appear evenly spaced on a plot with a log frequency axis. This is not to be confused with the log scale selection on plots.

Linear spacing: This selection will cause the analysis points to be spaced uniformly in frequency.

Current analysis # (Next analysis #) is the current analysis number. The main purpose of this is to distinguish waveforms from different analysis on plots--e.g., if you plot v(1) from analysis 2 and v(1) from analysis 3, they would be labeled as "v(1),2" and "v(1),3" on the plot. Clicking the "Reset" button sets the next analysis number to zero. When the analysis number is zero, waveforms have no analysis indicator (e.g., just plain "v(1)").

Analyze Internal T-line Voltages: Transmission line animation plot calculations (plots that are animated on the transmission lines) will be enabled if this is selected. This selection will cause a slower analysis and more disk space usage, so it should not be selected if you do not want to generate transmission line animation plots. This item must be selected in order to use the "Animate AC Animation Plots" feature under the "See Results" menu.

T-line points refers to the number of data points on each animation plot. Do not confuse this number with "Number of points" described above, which will determine the number of transmission line frames played when the animation is run (see "Animate AC Animation Plots" under the "See Results" menu). This number is relevent only if the "Enable Transmission line Analysis" item has been selected (see above). This number is limited to less than 1,500 points.

Reference resistance is the reference resistance used in the port analysis for calculating the s-parameters.

OK saves the analysis settings and returns to the circuit editor.

OK-Run AC Analysis saves the analysis settings and starts an ac analysis.

OK-Run Port Analysis saves the analysis settings and starts a port analysis.

4.5.2 Setup Transient Analysis

The Setup Transient Analysis dialog box is used to define the parameters of a transient analysis, including transmission line animation setup. The contents of the Setup Transient Analysis dialog box are detailed below. Changes made to item in the dialog box are applied to the next analysis.

Current analysis # (Next Analysis Number) is the current analysis number. The main purpose of this is to distinguish waveforms from different analysis on plots--e.g., if you plot v(1) from analysis 2 and v(1) from analysis 3, they would be labeled as "v(1),2" and "v(1),3" on the plot. Clicking the "Reset" button sets the next analysis number to zero. When the analysis number is zero, waveforms have no analysis indicator (e.g., just plain "v(1)").

Start time: This value is currently ignored. The transient analysis always starts at zero.

Stop time is the stop time of the transient analysis.

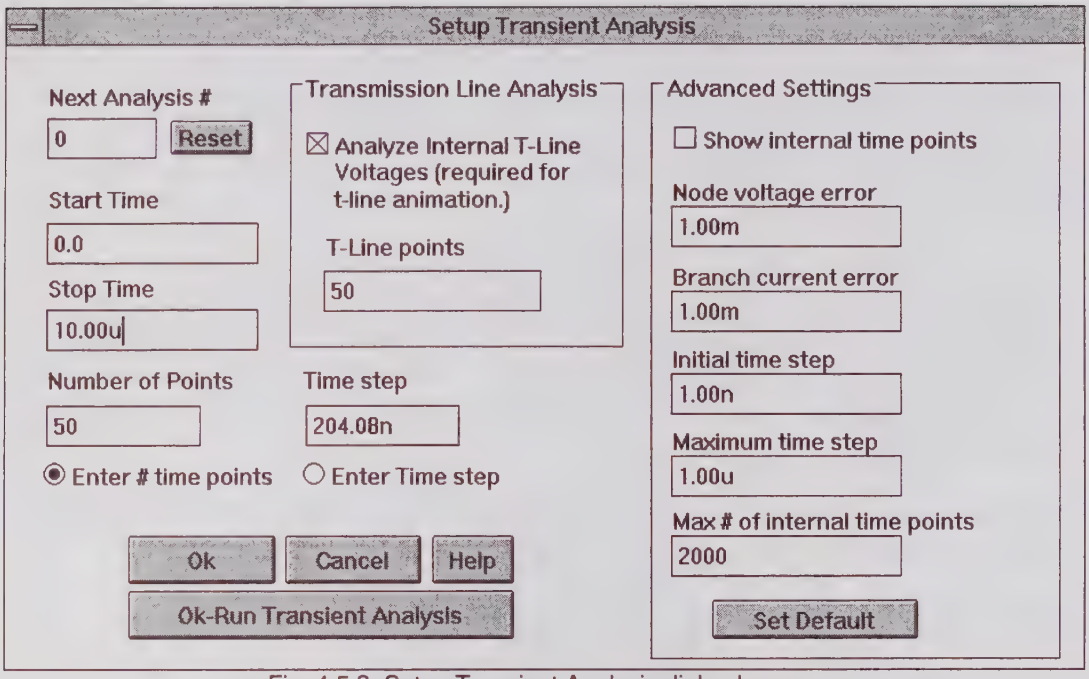


Fig. 4.5.3: Setup Transient Analysis dialog box

Number of points is the number of time points to be calculated in the transient analysis (and, therefore, to be plotted on transient plots). If you prefer, you can enter the time step between points instead (see the next items). See also "Show Internal Time Points" below. You may not have more than 15,000 points.

Time step is the time step between transient analysis data points. This is an alternative to entering the number of points.

Enter # time points: Select this button if you prefer to enter the number of points. The time step is automatically calculated.

Enter time step: Select this button if you prefer to enter the time step. The number of points is automatically calculated. If the stop time cannot be integrally divided by the time step, then the time step is also recalculated to the nearest possible value.

Analyze Internal T-line Voltages: Transmission line animation plot calculations (plots that are animated on the transmission lines) will be enabled if this is selected. This selection will cause a slower analysis and more disk space usage, so it should not be selected if you do not want to generate transmission line animation plots. This selection cannot be made simultaneously with the "Show Internal Time Points" selection (described below under "Advanced Settings"). This item must be selected in order to use the "Animate Transient Animation Plots" feature (under the "See Results" menu).

T-line points is the number of data points on each animation plot. Do not confuse this number with "Number of Points" described above, which will determine the number of transmission line frames played when the animation is run (see "Animate Transient Animation Plots" under the "See Results" menu). This number is relevent only if the "Enable Transmission Line Analysis" item has been selected (see above). This number is limited to less than 1,500 points.

Ok saves the analysis settings and returns to the circuit editor.

Ok-Run Transient Analysis saves the analysis settings and starts a transient analysis. See "Run Transient Analysis".

Advanced settings control the convergence and accuracy of the transient analysis. These items are recommended only for the experienced user.

Show internal time points: Selecting this button causes the "Number of Points" setting (see above) to be ignored. Instead, the internal time points of the analysis will be shown. In actuality, whenever a transient analysis is run, TRAVIS automatically determines where analysis time points need to go to prevent errors in the results (the allowable errors are specified as described below). TRAVIS additionally computes the waveform values at the user-requested uniformly spaced time points. As a result, checking this selection is actually a more efficient way of running the analysis since only the points that are actually needed will be calculated. It will result in nonuniform time point spacing in the analysis results and will eliminate any aliasing problems due to the user not choosing enough points. However, making this selection also precludes the transmission line analysis and, therefore, the ability to view these voltages with animation (see "Enable Transmission Line Analysis" above).

Node voltage error controls the maximum node voltage error allowed by the convergence algorithm. The larger this number is, the faster the analysis, but the less accurate it will be.

Branch current error controls the maximum branch current error allowed by the convergence algorithm. The larger this number is, the faster the analysis, but the less accurate it will be.

Initial time step is the initial time step used by TRAVIS. This is an initial guess for the time step control algorithm.

Maximum time step is the maximum time step used by TRAVIS. The time step control algorithm will never go longer than this amount of time to calculate a point in the analysis. By default, TRAVIS sets this number to 10% of the transient analysis stop time.

Maximum # time points is the maximum number of total time points allowed by TRAVIS. This includes internal time points generated plus the uniform time points requested. If you have checked the "Use Internal Time Points" box and this number is greater than 15,000, then it will be limited to 15,000 when the analysis is run.

Set default sets the default values for the advanced parameters.

4.5.3 Run DC Analysis

Selecting this menu item starts a dc analysis of the circuit. The Analysis Status dialog box pops up to display the progress of the analysis. If there are any errors in your circuit, an error message is displayed in this dialog box indicating what is wrong and (usually) what component has the problem. If you pop up the Edit Component dialog box of the erroneous component and look in the resolved information field of that dialog box, it will contain more precise detail on the problem with that component (you may have to scroll down to see the error information). Note also that more detail can be found on most errors by simply hitting the "F1" key after the error message is displayed.

4.5.4 Run AC Analysis

Selecting this menu item starts an ac analysis of the circuit using the ac analysis settings. The Analysis Status dialog box will pop up displaying the progress of the analysis. First, the progress of the program internally checking and connecting your circuit and any subcircuits is displayed. If there are any errors in your circuit, an error message is displayed in the analysis status dialog box indicating what is wrong and (usually) what component has the problem. If you pop up the Edit Component dialog box of the erroneous component and look in the resolved information field of that dialog box, it will contain more precise detail on the problem with that component (you may have to scroll down to see the error information). Note also that more detail can be found on most errors by simply hitting the "F1" key after the error message is displayed. If the circuit is OK, then the actual analysis is started. TRAVIS reports percentage done at approximately five-second intervals. The percentage done refers to how far in frequency the analysis has progressed (e.g., if your analysis goes from 0 to 20kHz, and you have selected linear frequency spacing, then when TRAVIS gets to 10kHz, it is 50% done).

At any time during the analysis, you can click the "Cancel" button to abort. When you do this, the analysis halts and displays some status information as well as the time spent simulating. The "Cancel" button then turns into an "OK" button. Click this "OK" button to close the analysis status dialog box. You can still click on circuit nodes and voltage sources to obtain partial analysis results calculated up until the point the analysis was canceled.

For information on seeing the analysis results, see "AC Rectangular Plot" and "AC Polar Plot". (see also the "Setup AC Analysis." menu item.)

4.5.5 Run Port Analysis

Selecting this menu item starts a port analysis of the circuit using the ac analysis settings. A dialog box will pop up displaying the progress of the analysis. First, the progress of the program internally checking and connecting your circuit and any subcircuits is displayed. If there are any errors in your circuit, an error message is displayed in the analysis status dialog box indicating what is wrong and (usually) what component has the problem. If you pop up the Edit Component dialog box of the erroneous component and look in the resolved information field of that dialog box, it will contain more precise detail on the problem with that component (you may have to scroll down to see the error information). Note also that more detail can be found on most errors by simply hitting the "F1" key after the error message is displayed. If the circuit is OK, then the actual analysis is started. TRAVIS reports percentage done at approximately five-second intervals. The percentage done refers to how far in frequency the analysis has progressed (e.g., if your analysis goes from 0 to 20kHz, and you have selected linear frequency spacing, then when TRAVIS gets to 10kHz, it is 50% done).

In the single-port case, a port analysis is just a special type of ac analysis. In the dual-port case, a port analysis is two separate ac analyses. For each port, TRAVIS internally adds a voltage source in series with a terminating resistor at that port, and adds just a terminating resistor at the other port (if it exists) and runs an ac analysis. The appropriate port parameters are calculated from these analyses when you go to see the results.

At any time during the analysis, you can click the "Cancel" button to abort. When you do this, the analysis halts and displays some status information as well as the time spent simulating. The "Cancel" button then turns into an "OK" button. Click this "OK" button to close the analysis status dialog box. You can still click on circuit nodes and voltage sources to obtain partial analysis results calculated up until the point the analysis was canceled.

For information on seeing the analysis results, see "Port Rectangular Plot" and "Port Polar Plot". (see also "Setup AC Analysis.")

4.5.6 Run Transient Analysis

Selecting this menu item starts a transient analysis of the circuit using the transient analysis settings. A dialog box will pop up displaying the progress of the analysis. First, the progress of the program internally checking and connecting your circuit and any subcircuits is displayed. If there are any errors in your circuit, an error message is displayed in the analysis status dialog box indicating what is wrong and (usually) what component has the problem. If you pop up the Edit Component dialog box of the erroneous component and look in the resolved information field of that dialog box, it will contain more detail on the problem with that component (you may have to scroll down to see the error information). Note also that more detail can be found on most errors by simply hitting the "F1" key after the error message is displayed. If the circuit is OK, then the actual analysis is started. TRAVIS reports percentage done at approximately

five-second intervals. The percentage done refers to how far in time the analysis has progressed (e.g., if your analysis goes from 0 to 10us, then when TRAVIS gets to 5us, it is 50% done).

At any time during the analysis, you can click the "Cancel" button to abort. When you do this, the analysis halts and displays some status information as well as the time spent simulating. The "Cancel" button then turns into an "OK" button. Click this "OK" button to close the analysis status dialog box. You can still click on circuit nodes and voltage sources to obtain partial analysis results calculated up until the point the analysis was canceled.

For information on seeing the analysis results, see "Transient plot". (See also "Setup Transient Analysis.")

4.5.7 Write Out Netlist

Selecting this menu item writes a simple circuit netlist to a file. This is useful for debugging your circuit if you are confused (or suspicious) about how your circuit is being connected. It also causes all mathematical expressions and function arguments to be evaluated so you can check for undefined variables or expression syntax errors, etc.

Chapter 5

Seeing Analysis Results

Once you have run an analysis, you are ready to look at the results of the analysis. Except for dc voltages, all analysis results can be sent to a plot window. There are menu items that control whether a waveform is sent to a plot that is already open or a new plot is created (see "Open New Plot" and "Use Active Plot" under the "See Results" menu). There are also menu items that control whether the plots go behind other windows or stick on top (see "Bring Plots to Top", "Stick All Plots on Top" and "Unstick All Plots from Top" under the "See Results" menu).

5.1 DC ANALYSIS RESULTS

Once you have run a dc analysis, you are ready to see the dc values of the circuit.

1. If the status text does not already show "Click wire/component for dc value", select "DC Value" from the "See Results" menu.
2. Press down the left mouse button on a wire. The name of the voltage on that wire pops down (e.g., v(5)) along with some other items (v(5)+, v(5)-, etc). Move the mouse pointer over the "v(5)" and release the left button. Now the voltage on that wire is attached to the mouse pointer.
3. Click the left mouse button to place the voltage.

For more information on seeing dc analysis results, see the description of the "dc Value" item under the "See Results" menu.

5.2 AC ANALYSIS RESULTS

Once you have run an ac analysis, you are ready to see plots of ac waveforms of the circuit.

1. If the status text does not already show "Click wire/component for ac rectangular plot", select "AC Plot->Rectangular (mag/phase or real/imag)" from the "See Results" menu. If you want to see a polar plot instead, select "AC Plot->Polar".
2. Press down the left mouse button on a wire. The name of the voltage on that wire pops down (e.g., v(5)) along with some other items (v(5)+, v(5)-, etc). Move the mouse pointer over the "v(5)" and release the left button. Now the voltage on that wire is sent to a plot window.

For more information on seeing ac analysis results, see the descriptions of the "AC Plot->Rectangular" and "AC Plot->Polar" items under the "See Results" menu. For more information on the operation of the plot window, see "Plot Windows."

5.3 PORT ANALYSIS RESULTS

Once you have run a port analysis, you are ready to see plots of scattering parameters of the circuit.

1. If the status text does not already show "Click wire/component for port polar plot", select "Port Plot->Polar" from the "See Results" menu. If you want to see a rectangular plot instead, select "Port Plot->Rectangular (mag/phase or real/imag)".
2. Press down the left mouse button on a port symbol. The names of the available s-parameters pop down (i.e., s11, s21, s12, s22 for two-port analysis or just s11 for a one-port analysis). Move the mouse pointer over the "s11" and release the left button. Now s11 of your circuit is sent to a plot window. For port analysis results, the smith graticule is the default.

For more information on seeing port analysis results, see the descriptions of the "Port Plot->Rectangular" and "Port Plot->Polar" items under the "See Results" menu. For more information on the operation of the plot window, see "Plot Windows."

5.4 TRANSIENT ANALYSIS RESULTS

Once you have run a transient analysis, you are ready to see plots of transient waveforms of the circuit.

1. If the status text does not already show "Click wire/component for transient plot", select "Transient Plot" from the "See Results" menu.
2. Press down the left mouse button on a wire. The name of the voltage on that wire pops down (e.g., v(5)) along with some other items (v(5)+, v(5)-, etc). Move the mouse pointer over the "v(5)" and release the left button. Now the voltage on that wire is sent to a plot window.

For more information on seeing transient analysis results, see the description of the "Transient Plot" item under the "See Results" menu. For more information on the operation of the plot window, see "Plot Windows."

5.5 TRANSMISSION LINE ANALYSIS RESULTS (ANIMATION)

The main feature of TRAVIS is the ability to display the waveforms in a transmission line over the time or frequency span of the analysis. For transient analysis, this means you can watch waveforms propagate along transmission lines and reflect at the ends. For ac analysis, this means you can observe how standing waves on the transmission line change over frequency.

Transient Transmission Line Analysis Results
The following step-by-step procedure will give you an introductory tour of the transient transmission line animation features of TRAVIS.

1. First put transmission line animation plots (also called animation plots for short) on the transmission lines you wish to study. For simplicity just select "Put Plots on All Transmission Lines" from the "See Results" menu. This will cause animation plots to appear on all of the transmission lines in the circuit window.
2. Select "Setup Transient Animation Plots" from the "See Results" menu. The Setup Transient Animation Plots dialog box pops up. Make sure the box next to "Total Voltage" is checked (this is the default.) Click the "Autoscale Animation Plot Limits" button. There is a pause while TRAVIS autoscales all of the animation plots. When it's done, click the "OK" button to close the dialog box.
3. Select "Animate Transient Animation Plots" from the "See Results" menu. This will pop up the Animation Control dialog box and rewind the transmission line animation.
4. If necessary, move the Animation Control dialog box so that it does not obscure the parts of your circuit you want to see during the animation.
5. Click the "Play>" button in the Animation Control dialog box. The transient transmission line animation starts. You can see the waveforms in the transmission lines as they reflect back and forth. For more detailed information on the Animation Control dialog box, see the description of the "Animate Transient Animation Plots" item under the "See Results" menu.
6. Now position the mouse pointer over a wire connected to one of the transmission lines. Press down the left mouse button and select "v(#)" to create a transient plot of that node's voltage.
7. Select the "Stick All Plots on Top" menu item from the "See Results" menu. This will make the plot window stick on top of the schematic so you can see it while you run the animation. Position and size this plot window so you can still see the animation plots of interest.

8. Click the "<<Rewind" button in the Animation Control dialog box, then click the "Play>" button again to start the animation again. Notice now that an "X" is tracking along the transient plot you popped up showing the spot on the transient plot that corresponds to the current animation time position. You can have any number of transient plot windows open at the same time, and all will have an "X" tracking the active trace during an animation.

9. Position the Animation Control dialog box so that it does not obscure the status text (in the upper left corner of the circuit window).

10. Position the mouse pointer over one of the animation plots. Notice that the coordinates of your mouse pointer relative to that plot are displayed in the status text.

11. Making sure that the circuit window has focus while the mouse pointer is over an animation plot, use the left-right cursor keys. The mouse pointer will snap to a waveform on the animation plot and the name of the waveform and its values are displayed in the status text. If you have multiple waveforms (forward or reverse, current or voltage, etc), select the up-down cursor keys to move to another waveform.

12. Press down the left mouse button over the animation plot. A menu pops up showing all of the transmission line waveforms available. Select one of them and release the mouse button. The waveform from the animation plot is sent to a plot window so you can get a closer look at it.

For more information, see the description of the "Animate Transient Animation Plots" item under the "See Results" menu.

AC Transmission Line Analysis Results:
The following step-by-step procedure will give you an introductory tour of the ac transmission line animation features of TRAVIS.

1. First put transmission line animation plots (also called animation plots for short) on the transmission lines you wish to study. For simplicity just select "Put Plots on All Transmission Lines" from the "See Results" menu. This will cause animation plots to appear on all of the transmission lines in the circuit window.

2. Select "Setup AC Animation Plots" from the "See Results" menu. The Setup AC Animation Plots dialog box pops up. Make sure the box in the "Total Voltage" row and "mag" column is checked (this is the default.) Click the "Autoscale Animation Plot Limits" button. There is a pause while TRAVIS autoscales all of the animation plots. When it's done, click the "OK" button to close the dialog box.

3. Select "Animate AC Animation Plots" from the "See Results" menu. This will pop up the Animation Control dialog box and rewind the transmission line animation.

4. If necessary, move the Animation Control dialog box so that it does not obscure the parts of your circuit you want to see during the animation.

5. Click the "Play>" button in the Animation Control dialog box. The ac transmission line animation starts. You can see the ac waveforms in the transmission lines as they change with frequency. For more detailed information on the Animation Control dialog box, see the description of the "Animate AC Animation Plots" item under the "See Results" menu.

6. Now position the mouse pointer over a wire connected to one of the transmission lines. Press down the left mouse button and select "v(#)" to create an ac plot of that node's voltage (polar or rectangular, it's your choice.)

7. Select the "Stick All Plots on Top" menu item from the "See Results" menu. This will make the plot window stick on top of the schematic so you can see it while you run the animation. Position and size this plot window so you can still see the animation plots of interest.

8. Click the "<<Rewind" button in the Animation Control dialog box, then click the "Play>" button again to start the animation again. Notice now that an "X" is tracking along the ac plot you popped up showing the spot on the transient plot that corresponds to the current animation time position. You can have any number of ac plot windows open at the same time, and all will have an "X" tracking the active trace during an animation.

9. Position the Animation Control dialog box so that it does not obscure the status text (in the upper left corner of the circuit window).

10. Position the mouse pointer over one of the animation plots. Notice that the coordinates of the mouse pointer relative to that plot are displayed in the status text.

11. Making sure that the circuit window has focus while the mouse pointer is over an animation plot, use the left-right cursor keys. The mouse pointer will snap to a waveform on the animation plot. The name of the waveform and its values are displayed in the status text. If you have multiple waveforms (forward or reverse, current or voltage, etc), select the up-down cursor keys to move to another waveform.

12. Press down the left mouse button over the animation plot. A menu pops up showing all of the transmission line waveforms available. Select one of them and release the mouse button. The waveform from the animation plot is sent to a plot window so you can get a closer look at it.

For more information, see the description of the "Animate AC Animation Plots" item under the "See Results" menu.

5.6 WAVEFORM MATH

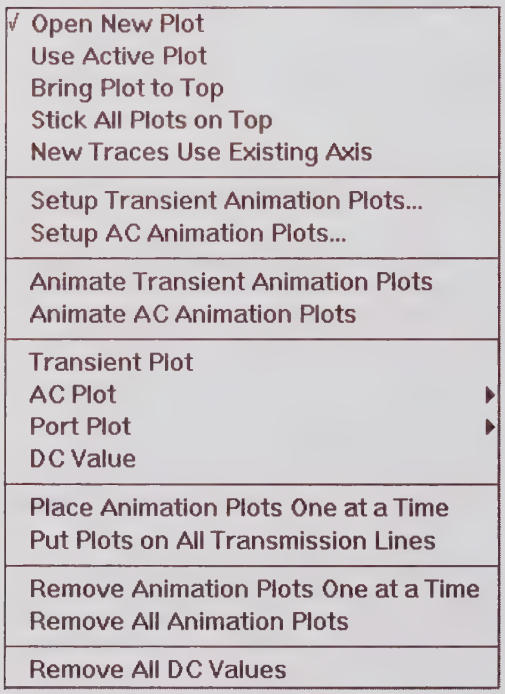
You can plot simple mathematical combinations of waveforms. For detailed information on what waveform math functions are available for each type of analysis and how to use

them, refer to the menu item description of the specific type of analysis results you are interested in doing math on under the "See Results" Menu:

Transient Plot
AC Plot->Rectangular
AC Plot->Polar
Port Plot->Polar
Port Plot->Rectangular

If you have plots that you would like to perform complex math functions on, you can write the plot waveforms to disk in an ascii file format and read it into another program that can perform more general arithmetic on arrays of numbers. For information on how to do this, see the description of the "Writing Selected Trace to File" item under the "Plot Window" menu.

5.7 SEE RESULTS MENU


√ Open New Plot
Use Active Plot
Bring Plot to Top
Stick All Plots on Top
New Traces Use Existing Axis
Setup Transient Animation Plots...
Setup AC Animation Plots...
Animate Transient Animation Plots
Animate AC Animation Plots
Transient Plot
AC Plot
Port Plot
DC Value
Place Animation Plots One at a Time
Put Plots on All Transmission Lines
Remove Animation Plots One at a Time
Remove All Animation Plots
Remove All DC Values
Fig. 5.7.1: See Results menu

For several of these menu items, once you select one, the program goes into the appropriate mode to allow the plotting of waveforms and the status text usually gives you a pretty good hint about what to do next. For example, when you select "Transient Plot", the status text will display "Click wire/component for transient plot". You can then

click on wires to get voltages and so on. See the description of each menu item for more information.

Once you have defined the waveform to be plotted, it is sent to the type of plot indicated by the menu item selected above. See "Plot Windows" for more information on the plots themselves.

The "See Results" menu items (of the circuit window) are described below.

5.7.1 Open New Plot

Selecting this menu item causes a new plot window to be opened whenever a new trace is plotted. The new trace is sent to the new plot window. This is a useful mode when you want to plot traces on separate plots.

See also the "Use Active Plot" menu item.

5.7.2 Use Active Plot

Selecting this menu item causes any new traces to be sent to the appropriate active plot (if it does not already exist, then a new plot window is opened). This is a useful feature when you want several traces on the same plot. The new traces either will cause a new y-axis to be created on the plot or will use the axis of a trace that is already on the plot (see "Plot Window: New Traces Use Existing Axis" for more information about this.)

How do you select which open plot to send the trace to? The trace is sent to the last plot window you clicked, which is known as the active plot and is indicated by the window caption. Each of the different types of plots will have an active plot--i.e., if you are in transient plot mode, waveforms will be sent to the active transient plot; if you are in ac polar plot mode, waveforms are sent to the active ac polar plot, and so on.

See also the "Open New Plot" menu item.

5.7.3 Bring Plots to Top

Selecting this menu item causes all existing plot windows (including ones that are iconized) to pop on top of the circuit editor.

For information on making plots stick on top see the "Stick on Top of Schematic" menu item under the "Plot" menu in the plot window, and the "Stick All Plot Windows on Top" menu item under the "See Results" menu.

5.7.4 Stick All Plots on Top

Selecting this menu item causes all existing plot windows (including ones that are iconized) to pop on top of the circuit editor and stick on top. It will also cause all future plot windows to have the property of sticking on top. A check mark next to this menu

item indicates that it has been selected. To unstick the plot windows, just reselect this menu item (and the check mark will disappear).

See the "Stick On Top of Schematic" menu item under the "Plot" menu in the plot window, which causes only the single plot window to stick on top.

5.7.5 New Traces Use Existing Axis

This option applies only when you have selected to send traces to active plots (see "Use Active Plot" under the "See Results" menu). Normally when a trace is sent to the active plot window, a new vertical axis is created for that trace. Selecting this menu item will cause the new trace to be put on the same axis as the active trace with the limitation that phase waveforms always use the right-hand axis, whereas all other waveforms use the left axis. A check mark appears next to this menu item to indicate it has been selected.

If there is an incompatibility between the new trace being plotted and the existing axis (i.e., the existing axis is a log scale, but the new trace has values less than zero), then a new axis is created for the new trace regardless of the "New Traces Use Existing Axis" setting. To undo this, select the menu item again and the check mark will disappear.

Because there is only one vertical and one horizontal axis on polar plots, this option does not apply to polar plots.

See the "New Traces Use Existing Axis" menu item under the "Plot" menu in the plot window, which causes only the single plot window to have this property.

5.7.6 Set Up Transient Animation Plots

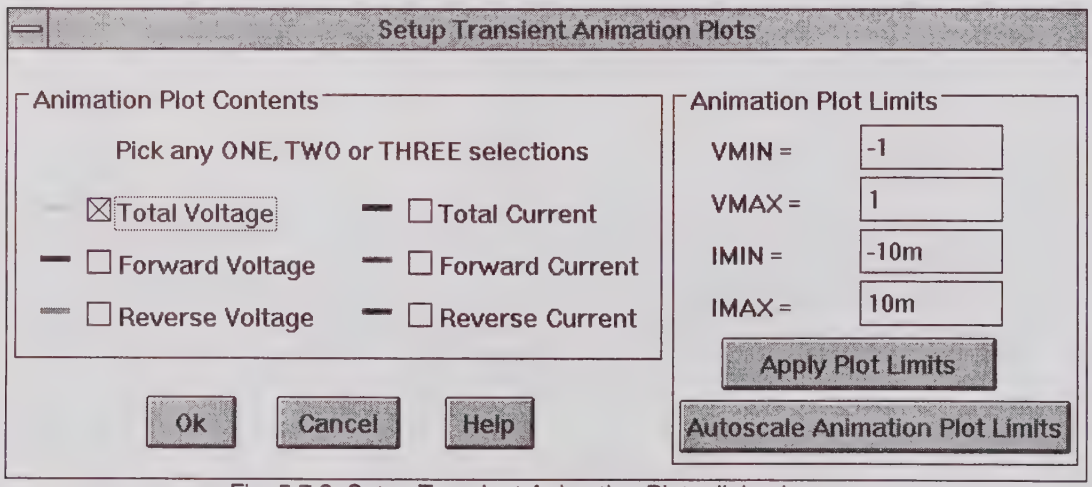


Fig. 5.7.2: Setup Transient Animation Plots dialog box

This dialog box contains the following items.

Animation Plot Contents: Select one, two, or three of these items to indicate what waveforms you want plotted in the animation plots. The waveforms you may have in the animation plots are listed below along with their respective colors.

Total voltage	yellow
Forward voltage	red
Reverse voltage	cyan
Total current	blue
Forward current	green
Reverse current	purple

Animation Plot Limits: This is a convenient way to set the animation plot limits for all of the transmission lines in the circuit window. Simply edit the values in the boxes and click the "Apply Plot Limits" button. All of the animation plot limits shown will be applied to all transmission lines in the circuit window (Note: The only transmission lines whose plot limits are affected are those in the circuit window that this dialog box was launched from.) (See also "Editing Transmission Lines" for information on the VMAX, VMIN, IMAX, and IMIN parameters.)

Autoscale Animation Plot Limits: If this button is clicked, then the transmission line voltages are autoscaled according to the current analysis results. VMAX of all of the transmission lines will be set to the maximum total voltage that actually occurred in all of the transmission lines; likewise with the VMIN, IMAX, and IMIN parameters. TRAVIS will autoscale only according to the waveforms you have selected to be in the animation plots--e.g., it won't autoscale IMAX and IMIN if you don't have any current waveforms selected for plotting.

The autoscale feature is convenient for quickly setting the plot limits on all of the transmission lines without knowing the general size of the waveforms ahead of time. You need to reclick this button after every analysis if you want to maintain this "best fit." Keep in mind that for large transient analysis (many time points or many T-line points or many transmission lines), the autoscaling operation gets more time consuming.

Besides the color of the waveforms, you can also see what signal each waveform on the transmission line represents by clicking on the circuit window that contains the animation plot (to make it active), positioning the mouse pointer over the animation plot of interest, then using the arrow keys to track the traces. The names of the traces will appear in the status bar in the upper left corner of the circuit window.

5.7.7 Setup AC Animation Plots

This dialog box contains the following items.

Animation Plot Contents: Select one, two, three, or four of these items to indicate what waveforms you want plotted in the animation plots. This also controls which waveforms you will be able to send to a transmission line plot window. Using the dis-

played array of check boxes, you can plot the magnitude, phase, real, and imaginary parts of the following waveforms (listed with their respective colors).

Total voltage	yellow
Forward voltage	red
Reverse voltage	cyan
Total current	blue
Forward current	green
Reverse current	purple

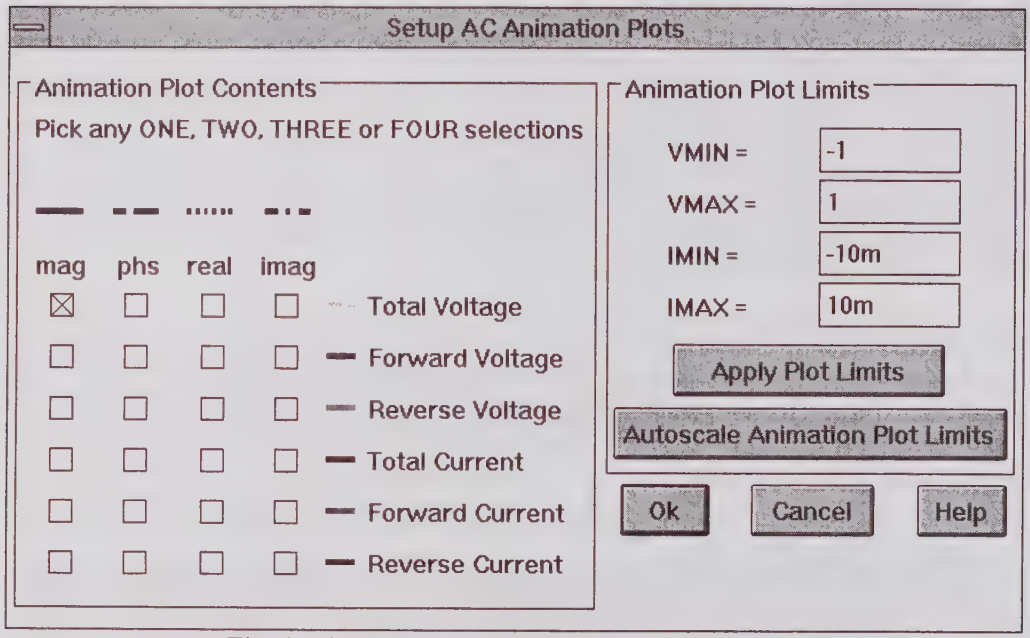


Fig. 5.7.3: Setup AC Animation Plots dialog box

The magnitude, phase, real, and imaginary parts of these waveforms are distinguished by different line styles as follows:

Magnitude	solid	————
Phase	dash	— — — —
Real	dot	- - - - - - - -
Imaginary	dash-dot	— · — · —

Animation Plot Limits: This is a convenient way to set the animation plot limits for all of the transmission lines in the circuit window. Simply edit the values in the boxes and click the "Apply Plot Limits" button. All of the animation plots limits shown will be applied to all transmission lines in the circuit window (Note: The only transmission lines whose plot limits are affected are those in the circuit window that this dialog box

was launched from.) (See also "Editing Transmission Lines" for information on the VMAX, VMIN, IMAX, and IMIN parameters.)

Autoscale Animation Plot Limits: If this button is clicked, then the transmission line voltages are autoscaled according to the current analysis results. VMAX of all of the transmission lines will be set to the maximum total voltage that actually occurred in all of the transmission lines; likewise with the VMIN, IMAX, and IMIN parameters. TRAVIS will autoscale only according to the waveforms you have selected to be in the animation plots--e.g., it won't autoscale IMAX and IMIN if you don't have any current waveforms selected for plotting.

The autoscale feature is convenient for quickly setting the plot limits on all of the transmission lines without knowing the general size of the waveforms ahead of time. You need to reclick this button after every analysis if you want to maintain this "best fit." Keep in mind that for large transient analysis (many time points or many T-line points or many transmission lines) the autoscaling operation gets more time consuming.

Besides the color of the waveforms, you can also see what signal each waveform on the transmission line represents by clicking on the circuit window that contains the animation plot (to make it active), positioning the mouse pointer over the animation plot of interest, then using the arrow keys to track the traces. The names of the traces will appear in the status bar in the upper left corner of the circuit window.

5.7.8 Animate Transient Animation Plots

After running a transient analysis with the "Analyze Internal T-line Voltages" box checked (see "Setup Transient Analysis"), you are ready to begin animation of the transient animation plots. Select the "Animate Transient Animation Plots" item in the "See Results" menu. This will pop up an Animation Control dialog box that contains some simple to use controls that allow manipulation of the animation plots. Next to each transmission line there is a small right-angle arrow symbol. This represents the origin and positive reference directions. The vertical arrow is the direction of positively increasing voltage or current and the horizontal arrow is the direction of forward current or voltage flow.

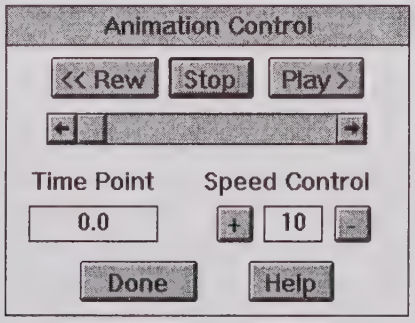


Fig. 5.7.4: Transient Animation Control dialog box

Transient Animation Control dialog box
The Transient Animation Control dialog box items are as follows:

<<Rew: This button rewinds the animation. The animation time point is set to "Start Time" (from the Setup Transient Analysis dialog box). Since the start time is only allowed to be zero, that is what the animation always rewinds to.

Stop: This button stops the animation at the current time point allowing you to examine the transmission line voltages at a certain point in time.

Play>: This starts the animation running from the current time point. The animation stops when the "Stop Time" (from Setup Transient Analysis dialog box) is reached.

Horizontal Scroll Bar: Use this scroll bar just as any other scroll bar. Clicking on the right arrow steps the animation forward one time point. Clicking the left arrow steps the animation backward one time point. Dragging the scroll button drags you quickly back or forward through the animation so you can jump to the time point of interest. Clicking to the right of the scroll button causes the animation to jump forward 10%. Clicking to the left of the scroll button causes the animation to jump backward 10%.

Time Point: This indicates the current time point of the animation. As you step forward in time (by pressing "Play>", for example) this number increases in increments equal to the "Time Step" value in the Setup Transient Analysis dialog box.

Speed Control: This controls how fast the analysis runs when the "Play>" button is pressed. The number can vary from 0 (slowest) to 10 (fastest). At 10, the animation goes as fast as possible (the faster your computer, the faster this will be). At 0, the program delays 1 second between time points. Each decrement of the speed number causes an additional 100ms delay between time points in the animation. This parameter does not affect the speed of the horizontal scroll bar left- and right-arrows, so if you want to quickly skip a part of the animation, you can click these buttons.

Several menu items are disabled during animation--basically any menu items related to editing the circuit or running an analysis. In addition, you are not allowed to open more than one animation control at a time. The one animation control will control animations for all animation plots in the main circuit window as well as any opened subcircuit windows.

When you click the "Play>" button in the animation control, the animation begins running and you can see the animation plots "flow" like a movie. You are seeing the current and voltage waveforms as they propagate and reflect within the transmission line. You can gain insight into the reasons the signals you observe on the ends of the transmission line look the way they do. For information on the contents and colors of the animation plots see "Setting up Transient Animation Plots."

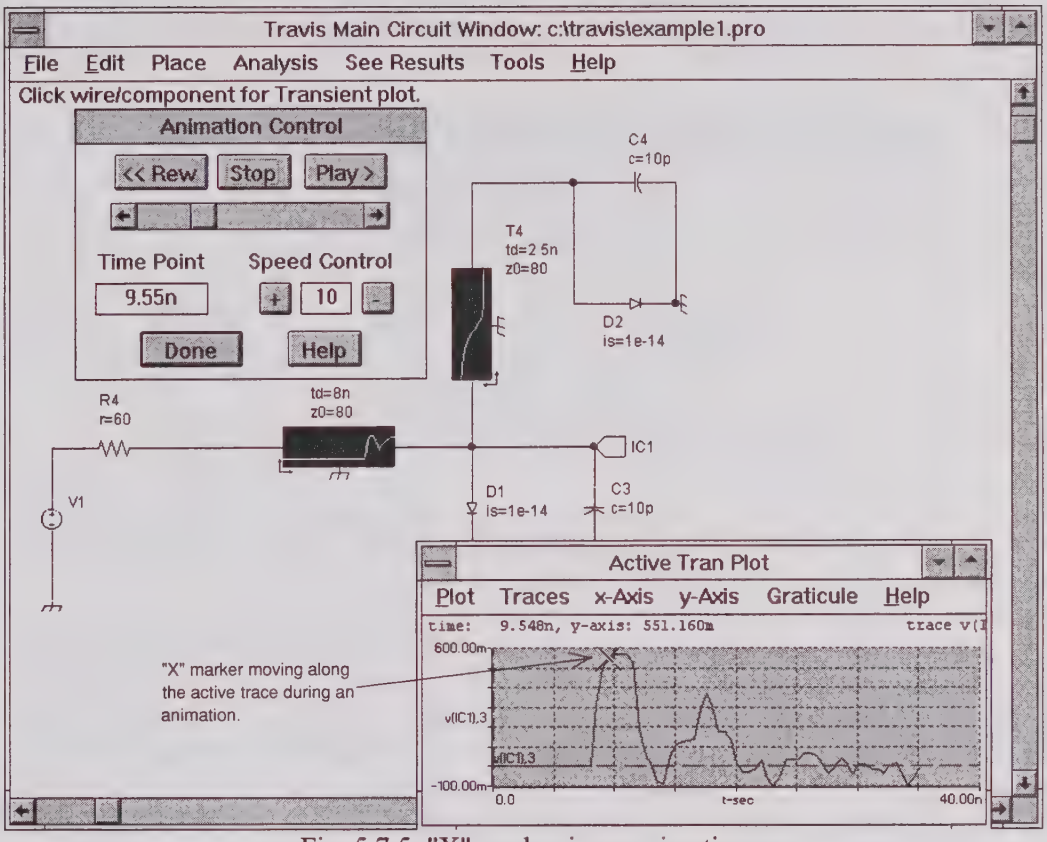


Fig. 5.7.5: "X" marker in an animation

In addition to the animation plots animating, all open transient plot windows will have an "X" marker moving along the active trace of the plot. This includes both standard transient plots (signal plotted versus time) and transient transfer plots (signal A versus signal B versus time). By following the "X" marker on the transient plots while watching the animation plot animations, you can see exactly how the internal transmission line voltages and currents correspond to the external signals of your circuit. (For more information on transient plots, see "Transient Plots" and "Plot Window.")

To view animation plots more closely, you can zoom in on them using the view scale menu item (under the "Edit" menu). The animation plots' actual resolution is higher than what is shown on the display, so zooming in on them will actually produce more visible detail. You can also send an animation plot to a plot window by pressing down the left mouse button over the animation plot. A menu pops up with a list of the waveforms available to be plotted (see the "Setup Transient Animation Plots" menu item for information on setting which transmission line waveforms are available for plotting). Select the item you want and release the mouse button. A plot window opens up with the waveform--the horizontal axis is in units of time that correspond to the transmission line propagation time starting from the origin of the transmission line. You can now

closely examine the waveform or send it to the printer. This feature is available only while the animation control is on.

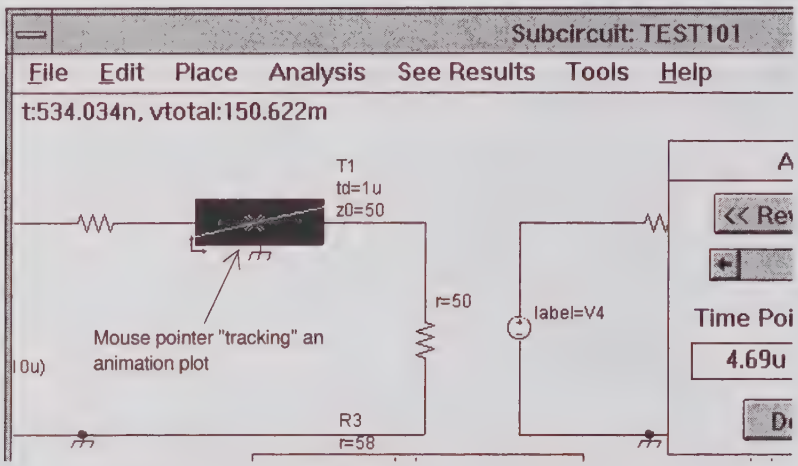


Fig. 5.7.6: Trace tracking an animation plot

You can see signal values on the animation plots by running the mouse pointer over them when the circuit window is the active window. When you do this, the status bar of the circuit window will start displaying the coordinates of your mouse pointer within that plot. In addition you can trace track along the animation plot traces using the arrow keys the same way you do when you are tracking along traces in a standard plot window. Make sure the circuit window with the animation plots of interest is the active window, move the mouse pointer over the animation plot, and use the arrow keys to start tracking.

All of the above-mentioned features as well as any other circuit features not disabled can be used at any time during an animation regardless of whether the animation is actually running or stopped. This is useful if, for example, you have a large circuit and want to follow a pulse as it travels through the circuit. You can click the circuit window scroll bars and scroll the circuit while the animation is running.

You must have run a transient analysis to see these results. In addition, you must have enabled the "Analyze T-line Voltages" feature in the Setup Transient Analysis Control dialog box before running the analysis.

5.7.9 Animate AC Animation Plots

After running an ac analysis with the "Analyze Internal T-line Voltages" box checked (see "Setup AC Analysis"), you are ready to begin animation of the ac animation plots. Select the "Animate AC Animation Plots" item under the "See Results" menu. This will pop up an Animation Control dialog box that contains some simple to use controls that allow manipulation of the animation plots. Next to each transmission line there is a small right-angle arrow symbol. This represents the origin and positive reference direc-

tions. The vertical arrow is the direction of positively increasing voltage or current and the horizontal arrow is the direction of forward current or voltage flow.

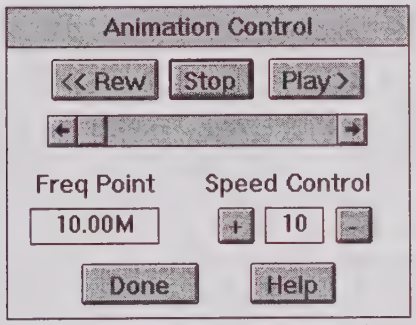


Fig. 5.7.7: AC Animation Control dialog box

AC Animation Control dialog box
The AC Animation Control dialog box items are as follows:

<<Rew: This button rewinds the animation. The animation frequency point is set to "Start Frequency" (from the Setup AC Analysis dialog box).

Stop: This button stops the animation at the current frequency point allowing you to examine the transmission line voltages at a certain point in frequency.

Play>: This starts the animation running from the current frequency point. The animation stops when the "Stop Frequency" (from Setup AC Analysis dialog box) is reached.

Horizontal Scroll Bar: Use this scroll bar just as any other scroll bar. Clicking on the right arrow steps the animation forward one frequency point. Clicking the left arrow steps the animation backward one frequency point. Dragging the scroll button drags you quickly back or forward through the animation so you can jump to the frequency point of interest. Clicking to the right of the scroll button causes the animation to jump forward 10%. Clicking to the left of the scroll button causes the animation to jump backward 10%.

Frequency Point: This indicates the current frequency point of the animation. As you step forward in frequency (by pressing "Play>", for example) this number increases in frequency increments. These increments are uniform if you checked the "Linear Spacing" box in the Setup AC Analysis dialog box (see the "Setup AC Analysis" item under the "Analysis" menu). Otherwise they are logorithmic.

Speed Control: This controls how fast the analysis runs when the "Play>" button is pressed. The number can vary from 0 (slowest) to 10 (fastest). At 10, the animation goes as fast as possible (the faster your computer, the faster this will be). At 0, the program delays 1 second between frequency points. Each decrement of the speed number

causes an additional 100ms delay between frequency points in the animation. This parameter does not affect the speed of the horizontal scroll bar left- and right-arrows, so if you want to quickly skip a part of the animation, you can click these buttons.

Several menu items are disabled during animation--basically any menu items related to editing the circuit or running an analysis. In addition, you are not allowed to open more than one animation control at a time. The one animation control will control animations for all animation plots in the main circuit window as well as any opened subcircuit windows.

When you click the "Play>" button in the animation control, the animation begins running and you can see the animation plots "flow" like a movie. You are seeing the ac current and voltage waveforms as they reflect and form standing waves within the transmission line. You can gain insight into the reasons the signals you observe on the ends of the transmission line look the way they do. For information on the appearance and colors of the animation plots, see "Setup AC Animation Plots" (under the "See Results" menu).

In addition to the animation plots animating, all open ac plot windows (rectangular and polar) will have an "X" marker moving along the active trace of the plot. By following the "X" marker on the ac plots while watching the animation plot animations, you can see exactly how the internal transmission line voltages and currents correspond to the external signals of your circuit. (For more information on ac plots, see "AC Plot->Rectangular", "AC Plot->Polar" and "Plot Window.")

To view animation plots more closely, you can zoom in on them using the view scale menu item (under the "Edit" menu of the circuit window). The animation plots' actual resolution is higher than what is shown on the display, so zooming in on them will actually produce more visible detail. You can also send an animation plot to a plot window by pressing down the left mouse button over the animation plot. A menu pops up with a list of the waveforms available to be plotted (see "Setting Up AC Animation Plots" for information on setting which transmission line waveforms are available for plotting). Select the item you want and release the mouse button. A plot window opens up with the waveform--the horizontal axis is in units of time that correspond to the transmission line propagation time starting from the origin of the transmission line. You can now closely examine the waveform or send it to the printer. This feature is available only while the animation control is on.

You can see signal values on the animation plots by running the mouse pointer over them when the circuit window is the active window. When you do this, the status bar will start displaying the coordinates of your mouse pointer within that plot. In addition you can trace track along the animation plot traces using the arrow keys the same way you do when you are tracking along traces in a standard plot window. Make sure the circuit window with the animation plots of interest is the active window, move the mouse pointer over the animation plot, and use the arrow keys to start tracking.

All of the above-mentioned features as well as any other circuit features not disabled can be used at any time during an animation regardless of whether the animation is actually running or stopped.

You must have run an ac analysis to see these results. In addition, you must have enabled the "Analyze T-line Voltages" feature in the Setup AC Analysis Control dialog box before running the analysis.

5.7.10 Transient Plot

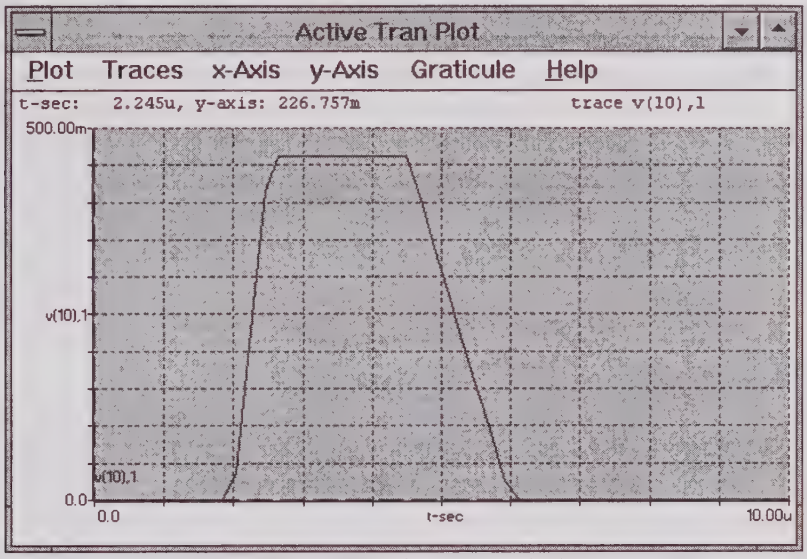


Fig. 5.7.8: Transient plot window

To see plots from a transient analysis, select the "Transient Plot" item from the "See Results" menu. The status text will display "Click wire/component for transient plot." Next, push down the left mouse button on a wire and a floating menu will pop up with six choices: v(#), v(#)-, v(#)/, v(#)+, v(#)*, v(#)vs., where "#" refers to the node name of the wire. If you select v(#), the transient voltage is sent to a transient plot.

The next four selections allow simple waveform math to be performed. For example, if you were to click on a wire and select "v(1)-" and then click on another wire and select "v(2)", then the difference voltage, v(1)-v(2), would be sent to a transient plot. You can do the same for the other math selections. TRAVIS allows up to four waveforms to be mathematically combined like this. The order of the mathematical operators is first come, first served.

The last selection, "v(#)vs.", is used when you want to plot one waveform versus another (called a "transfer" plot). For example, if you were to select "v(1)vs." on one node (wire) and then "v(2)" on another node, TRAVIS would plot a transfer curve of v(1)vs.v(2). (Note: After you have selected "v(#)vs.", TRAVIS allows only v(#) on the next node you click.) Since v(1) and v(2) are both transient plots, the v(1)vs.v(2) plot will show this transfer curve at the time points that were calculated in the transient analysis. If you trace track on this plot, the trace track value will display three values, v(1), v(2), and the time at which the current point occured in the transient analysis. As a

result, you can see where in the transfer curve the circuit is at any time point of the transient analysis.

When you press the left mouse button on a voltage source, dependent voltage source, or current meter, the transient current flowing through that component will pop up on a floating menu with the same six choices as above.

When you press the left mouse button on a volt meter, the voltage across the volt meter pops up on a floating menu with the same six choices as above. This allows waveform math on differential voltages such as the gain of an op-amp--e.g., v(out)/(v(pos)-v(neg))--in this case, you'd have a volt meter placed across the inputs to the op-amp.

You must have run a transient analysis to see these results. See "Setup Transient Analysis" and "Run Transient Analysis."

see Plot Windows for more information on the format and control of the plot window. See "Open New Plot" and "Use Active Plot" for information about which plot window the trace is sent to.

5.7.11 AC Plot->Rectangular (Mag/Phase or Real/Imag)

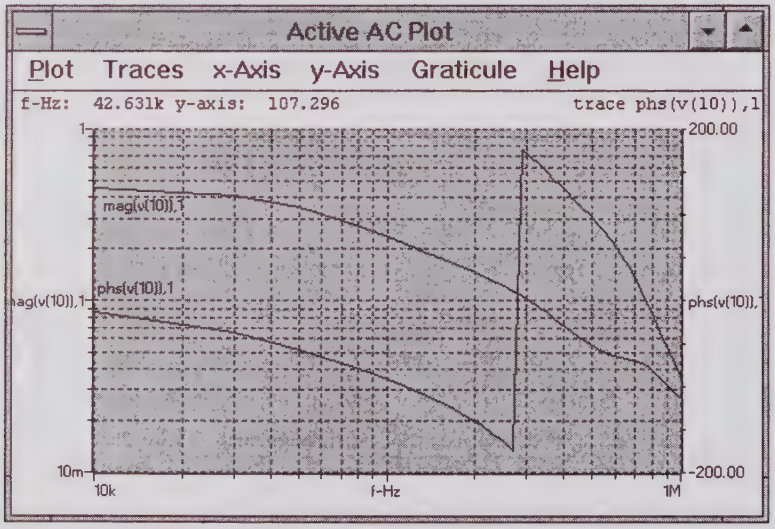


Fig. 5.7.9: Ac rectangular plot (log scale)

To see rectangular plots from an ac analysis, select either "AC Plot->Rectangular-Mag/Phase" or "AC Plot->Rectangular-Real/Imag" item from the "See Results" menu. Selecting "AC Plot->Rectangular-Mag/Phase" will cause the magnitude and phase of the waveform to be plotted. Selecting "AC Plot->Rectangular-Real/Imag" will cause the real and imaginary parts of the waveform to be plotted.

Selecting either of these menu items will cause the status text to display "Click wire/component for ac rectangular plot." Next, push down the left mouse button on a wire and a floating menu will pop up with five choices: v(#), v(#)-, v(#)/, v(#)+, v(#)*, where "#" refers to the node name of the wire. If you select v(#), the ac voltage is sent

to the active ac rectangular plot. The other selections allow simple waveform math to be performed. For example, if you were to click on a wire and select "v(1)-" and then click on another wire and select "v(2)", then the difference voltage, v(1)-v(2), would be sent to the ac active plot. You can do the same for the other math selections. TRAVIS allows up to four waveforms to be mathematically combined like this. The order of the mathematical operators is first come, first served.

When you press the left mouse button on a voltage source, dependent voltage source, or current meter, the ac current flowing through that component will pop up on a floating menu with the same five choices as above. Voltages and currents can be mathematically combined, for example, to obtain input resistance.

When you press the left mouse button on a volt meter, the voltage across the volt meter pops up on a floating menu with the same five choices as above. This allows waveform math on differential voltages such as the gain of an op-amp--e.g., v(out)/(v(pos)-v(neg)). In this case, you'd have a volt meter placed across the input to the op-amp.

You must have run an ac analysis to see these results. See "Setup AC Analysis" and "Run AC Analysis."

See "Plot Windows" for more information on the format and control of the plot window. See "Open New Plot" and "Use Active Plot" for information about which plot window the trace is sent to.

5.7.12 AC-Plot->Polar

To see polar plots from an ac analysis, select the "AC Plot->Polar" item from the "See Results" menu. The status text will display "Click wire/component for ac polar plot." Next, push down the left mouse button on a wire and a floating menu will pop up with five choices: v(#), v(#)-, v(#)/, v(#)+, v(#)*, where "#" refers to the node name of the wire. If you select v(#), the ac voltage is sent to the active ac polar plot. The other selections allow simple waveform math to be performed. For example, if you were to click on a wire and select "v(1)-" and then click on another wire and select "v(2)", then the difference voltage, v(1)-v(2), would be sent to the ac active plot. You can do the same for the other math selections. The program allows up to four waveforms to be mathematically combined like this. The order of the mathematical operators is first come, first served.

When you press the left mouse button on a voltage source, dependent voltage source, or current meter, the ac current flowing through that component will pop up on a floating menu with the same five choices as above. Voltages and currents can be mathematically combined, for example, to obtain input resistance.

When you press the left mouse button on a volt meter, the voltage across the volt meter pops up on a floating menu with the same five choices as above. This allows waveform math on differential voltages such as the gain of an op-amp--i.e., v(out)/(v(pos)-v(neg)). In this case, you'd have a volt meter placed across the input to the op-amp.

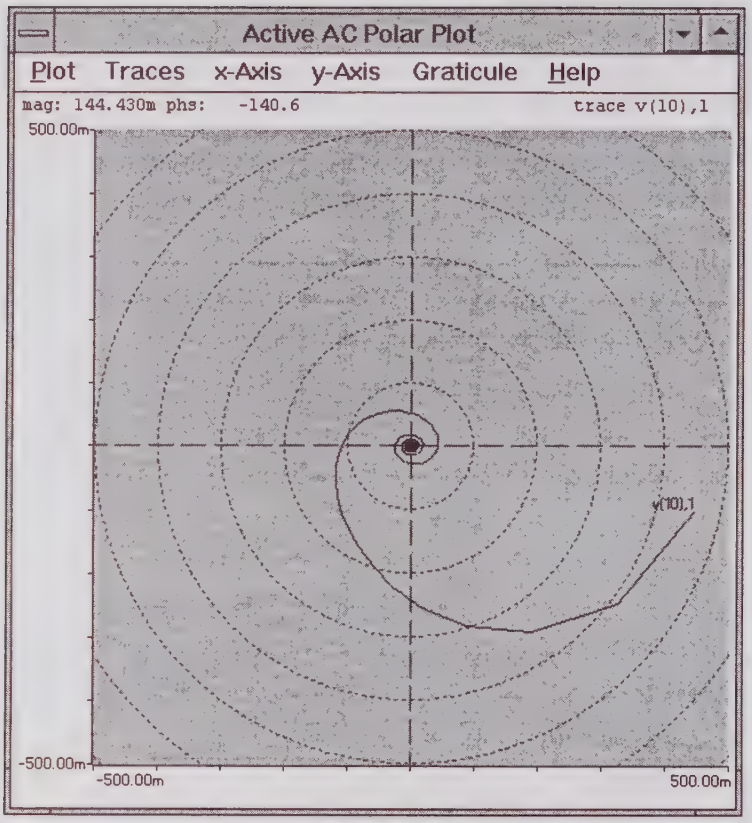


Fig. 5.7.10: Ac polar plot

You must have run an ac analysis to see these results. See "Setup AC Analysis" and "Run AC Analysis."

See "Plot Windows" for more information on the format and control of the plot window. See "Open New Plot" and "Use Active Plot" for information about which plot window the trace is sent to.

5.7.13 Port Plot->Polar

To see polar plots from a port analysis, select the "Port Plot->Polar" item from the "See Results" menu. The status text will display "Click wire/component for port polar plot." While in this mode, whenever a you push down the left mouse button on a port symbol, a floating menu pops down allowing the selection of an s-parameter. Select one of the s-parameters and release the mouse button. The selected s-parameter will be sent to a polar plot (with the Smith graticule turned on).

You must have run a port analysis to see these results. (See "Setup AC Analysis" and "Run Port Analysis.")

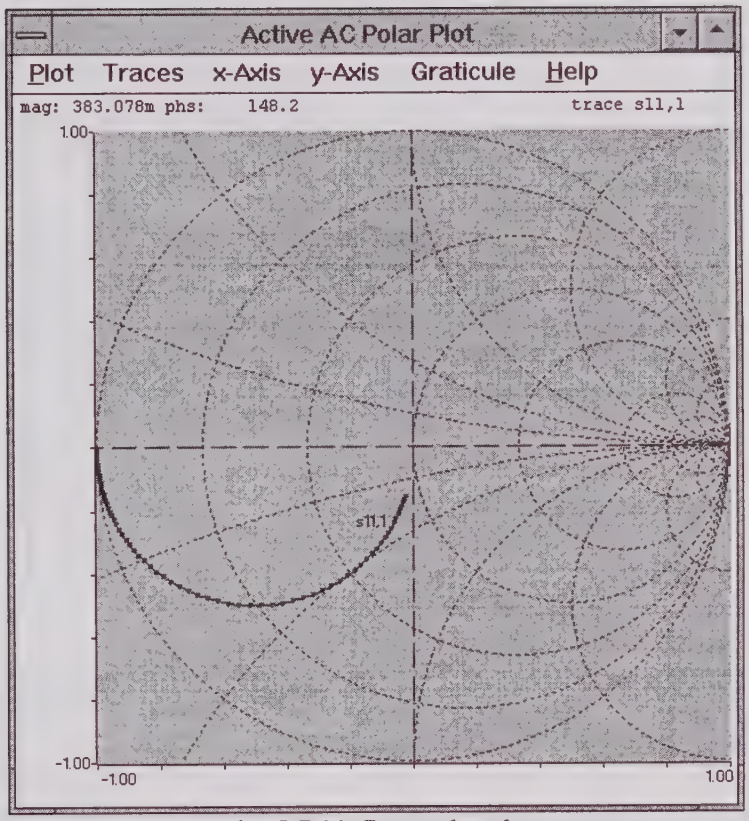


Fig. 5.7.11: Port polar plot

See "Plot Windows" for more information on the format and control of the plot. See "Open New Plot" and "Use Active Plot" for information about which plot window the trace is sent to.

5.7.14 Port Plot->Rectangular (Mag/Phase or Real/Imag)

To see rectangular plots from a port analysis, select either the "Port Plot->Rectangular-Mag/Phase" or "Port Plot->Rectangular-Real/Imag" item from the "See Results" menu. Selecting "Port Plot->Rectangular-Mag/Phase" will cause the magnitude and phase of the waveform to be plotted. Selecting "Port Plot->Rectangular-Real/Imag" will cause the real and imaginary parts of the waveform to be plotted.

Selecting either of these selections will cause the status text to display "Click wire/component for port rectangular plot." While in this mode, whenever you press down the left mouse button on a port symbol, a floating menu pops down allowing the selection of an s-parameter. Select one of the s-parameters and release the mouse button. The selected s-parameter will be sent to a rectangular plot.

You must have run a port analysis to see these results. See "Setup AC Analysis" and "Run Port Analysis."

See "Plot Windows" for more information on the format and control of the plot. See "Open New Plot" and "Use Active Plot" for information about which plot window the trace is sent to.

5.7.15 DC Value

To see a dc value from a dc analysis, select the "DC value" item from the "See Results" menu. The status text will display "Click wire/component for dc value." Now, when you push down the left mouse button on a wire, a floating menu will pop up with five choices: v(#), v(#)-, v(#)/, v(#)+, v(#)*, where "#" refers to the node name of the wire. If you select v(#), the dc voltage is attached to the mouse pointer. Click the left mouse button to place it.

The next four selections allow simple math to be performed. For example, if you click on a wire and select "v(1)-" and then click on another wire and select "v(2)", then the difference voltage, v(1)-v(2), is computed and attached to the mouse pointer. You can do the same for the other math selections. TRAVIS allows up to four selections to be mathematically combined like this. The order of the mathematical operators is first come, first served.

When you press the left mouse button on a voltage source, dependent voltage source, or current meter, the dc current flowing through that component will pop up on a floating menu with the same five choices as above.

When you press the left mouse button on a voltmeter, the dc voltage across the voltmeter pops up on a floating menu with the same five choices as above.

Select the "Remove All DC Voltages" menu item to remove the voltages you have placed.

You must have run a dc analysis to see these results. See "Run DC Analysis."

5.7.16 Place Animation Plots One at a Time

This menu selection causes the status text to display "Click t-lines to turn on animation plots." In this mode, an animation plot will appear on any transmission line that is clicked. This can be done before or after the analysis.

5.7.17 Put Plots on All Transmission Lines

This menu selection causes animation plots to be put on all transmission lines in the circuit editor window. This does not affect the analysis results and can be done before or after the analysis.

5.7.18 Remove Animation Plots One At a Time

This menu selection causes the status text to display "Click t-lines to turn off animation plots." In this mode, an animation plot will be removed from any transmission line that is clicked. This can be done before or after the analysis.

5.7.19 Remove All Animation Plots

This menu selection removes all animation plots from the circuit schematic. This has no effect on the analysis results and can be done before or after the analysis.

5.7.20 Remove All DC Values

This menu selection removes all dc values from the circuit schematic.

5.8 PLOT WINDOWS

There are two general types of plots: rectangular and polar. The most common examples of rectangular plots are seen when displaying transient analysis results, in which case the plots display a waveform versus time, and ac and port analysis results, in which case the plots will display a waveform versus frequency. Polar plots, on the other hand, plot the real part versus the imaginary part of a waveform (used only for ac and port analysis results). Rectangular plots are also used for plotting one transient waveform versus another. The "Transmission Line Designer" tool also uses rectangular plots to plot one transmission line parameter versus another.

There are a few items in the plot window worth noting. In the upper left corner of the plot window (under the "Plot" menu) is the track value. This displays the plot value under the current mouse pointer location (and, in some cases, may also display a message regarding that data point).

To the right of the track value is displayed the active trace. The active trace is the trace that is used as a reference in displaying the track value (e.g., if the active trace is volts versus time, then the track value units are volts and time; if the active trace is current versus time, then the track value units are amps and time). The active trace is also the trace that the crosshair moves along when a transmission line animation is in progress (see "Animate Transient Animation Plots" and "Animate AC Animation Plots").

The plot axes can be on the left, right, and bottom of the plot. For rectangular plots, voltage and current magnitude axis are placed on the left; the phase axis is placed on the right; and the frequency or time axis is placed on the bottom. For polar plots, only a single vertical and horizontal axis are allowed--these axis are marked off in rectangular coordinates. Rectangular plots may have log or linear axis (for ac and port analysis only).

For rectangular plots, the plot graticule can be on or off and log or linear, depending on the axis. Unlike the axis, however, there can be only one graticule at a time--you may select which trace the graticule is associated with. For polar plots, the plot graticule can be a standard polar or Smith graticule (the Smith graticule is simply a few constant impedance contours drawn on the plot). Graticule lines that are at zero (0 volts, 0 amps, 0 sec, etc.) are a different color (black) than the other graticule lines (blue).

The plot itself may contain one or more traces. Each trace is associated with a vertical and horizontal axis (all traces share one horizontal axis and in polar plots they share the vertical axis as well). Each trace also has a label (see the "Labels On" menu item under the "Traces" menu), which is placed at the beginning of the trace (e.g., at the earliest time point or lowest frequency point). Clicking the left button on a trace or its label causes that trace to turn red (i.e., the trace becomes selected) and also turns its associated vertical axis red. If you click on the trace label, the label attaches to the mouse pointer, allowing you to move it around with the mouse or the arrow keys on the keyboard. Clicking a second time puts the label down. Clicking on the right button causes all traces to become unselected. When a trace is selected or deselected (by clicking on it or its label), it becomes the active trace. Most of the plot window menu features act on the selected trace(s).

You can zoom in on any portion of the plot simply by pressing down and holding the left mouse button and dragging the mouse. A selection box is drawn indicating the region of the plot that will be expanded to fill the plot window. Releasing the left mouse button causes the plot to zoom. All traces, graticules, and axes are resized and rescaled to fit on the rescaled plot. Other scaling and resizing features are described under the "Plot" menu descriptions.

Trace tracking is accessed from the keyboard. If you hit the left or right arrow keys, the mouse pointer snaps to the nearest data point on the active trace and starts tracking along the trace displaying the actual trace data in the track value display in the upper left corner of the plot window. The right arrow key causes the pointer to advance on the waveform (move to higher frequencies or later times); the left arrow key causes the pointer to go the opposite direction.

The up and down arrow keys cause the pointer to hop to the next trace (waveform). Notice that the active trace changes to indicate this change. When the pointer hops to another waveform, it always hops to the same horizontal location. In other words, if the pointer was tracking at 4kHz on one trace, it will still be at 4kHz when it hops to another trace. This applies to polar as well as rectangular plots. If plots are displayed from different analysis runs that have different number of points or different point spacing, the cursor will find the nearest horizontal point. All keyboard-related features are active only in the active window.

A special type of rectangular plot is the transfer plot. This is available only for transient analysis results. In a transfer plot, one waveform is plotted versus another. All transient plot features apply to this type of plot. The trace tracking feature behaves similarly except that three values are displayed in the track value in the upper left corner: the vertical axis, the horizontal axis, and the time point. As such, the trace track value

shows where on the transfer curve the circuit is at any given time point in the transient analysis. For more information on this type of plot, see the "Transient Plot" item under the "See Results" menu.

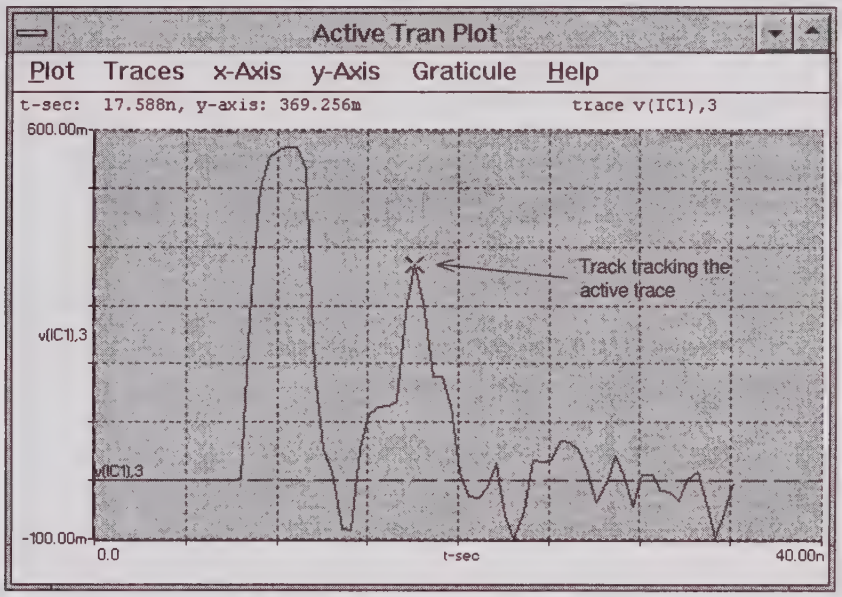


Fig. 5.7.12: Trace Tracking

5.8.1 Plot Window Menus

The plot window menus are as follows: (Note: in a polar plot, the x-axis corresponds to the real axis and the y-axis corresponds to the imaginary axis.)

Plots Menu

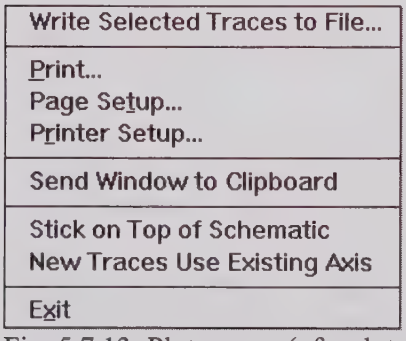


Fig. 5.7.13: Plots menu (of a plot window)

Write Selected Traces to File

This menu selection writes all selected traces to an ASCII file in column format. Multiple traces are written one after another in the file with labeling at the beginning of each trace. The data can be read into another software package from this file when the labels in the file are either removed or commented out in a way that the software package can handle. Select traces simply by clicking on them (selected traces are indicated in red--see "Plot Windows" for more information.)

Format of the file written to disk:

For rectangular plots there are two columns. The first column contains the x-axis variable (frequency or time), and the second column is the y-axis variable (voltage, current, phase, etc.).

For polar plots there are three columns. The first column contains frequency (all polar plots result from ac or port analysis). The second column contains the magnitude of the trace. The third column contains the phase of the trace in degrees. All columns are labeled.

Print

Selecting this menu item causes the plot to be sent to the printer using the current printer and page settings. A time and date stamp are added to the top of the plot.

Page Setup

Selecting this menu item causes a scaled representation of the printer page to pop up showing how the circuit will look when printed. Selecting "Zoom in" or "Zoom out" from the "Change Scaling" menu adjusts the magnification of the circuit diagram. The scroll bars can be used to adjust the positioning of the plot on the page.

When you resize this window, it automatically readjusts itself to correspond to the aspect ratio of the printed page. You should setup the printer (by selecting the "Printer Setup" menu item) before selecting this menu item as setting up the printer affects the page aspect ratio and, in general, how the plot will look on the printed page.

To quit from this window and save the current page setup, just close the window by double-clicking the upper left "System Menu" button of the window.

Printer Setup

When you select this menu item, the standard printer setup dialog box for your printer will pop up asking for various pieces of information such as the resolution and page orientation (and any other settings your particular printer uses).

The settings you select here are kept local to TRAVIS and will affect neither the system printer settings (in the windows control panel) nor the printer settings of other programs you may currently have running.

Send Window to Clipboard

Selecting this menu item causes the plot as is to be sent to the Windows clipboard in the form of a bit map. It can then be pasted into any other program that accepts bit maps. The background of the plot becomes white when it is sent to the clipboard to improve appearance when printed on paper.

Stick on Top of Schematic
Normally, the plot window will be overlapped (covered up) by the circuit window when you click on the circuit window. Selecting this menu item will cause this plot window to stick on top--i.e., the circuit window will stay underneath it even if you click on the circuit window. Selecting this menu item again causes the plot window to go back to its original behavior.

This feature is necessary if you want to simultaneously view the animation plots and the marker moving along the transient plots. (See also the "Animate Transient Animation Plots" and "Animate AC Animation Plots" menu items.)

New Traces Use Existing Axis--Rectangular Plots Only
This option applies only when you have selected to send traces to active plots (see "Use Active Plot" in the "See Results" menu). Normally when a trace is sent to an active plot window, a new vertical axis is created for that trace. Selecting this menu item will cause the new trace to be put on the same axis as the active trace.

If there is an incompatibility between the new trace being plotted and the existing axis (i.e., the existing axis is a log scale, but the new trace has values less than zero), then a new axis is created for the new trace regardless of the "New Traces Use Existing Axis" setting.

Because there is only one vertical and one horizontal axis on polar plots, this option is not available on polar plots.

Traces Menu

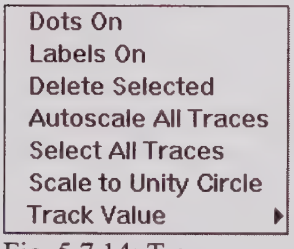


Fig. 5.7.14: Traces menu (of a plot window)

Dots On
Selecting this menu item causes the actual data points in the traces to become visible. Selecting it again turns the points off again.

Labels On
Selecting this menu item turns off the trace labels. Selecting it again turns the labels back on.

Delete Selected
All selected traces are deleted when this menu item is selected.

Autoscale All Traces
All traces in the plot are autoscaled to their respective axis. Nice plot limits and division sizes are automatically determined and applied. In polar plots, the origin is always put at the center of the plot and the horizontal and vertical scales are always set equal in size.

Scale to Unity Circle--polar plots only
This simply causes a polar plot to be scaled such that 0,0 is in the center of the plot and the real and imaginary axis limits are +1 to -1. Although you are allowed to do this with any polar plot, its main use is with s-parameters on a Smith graticule.

Track Value--polar plots only
There are four ways to display the trace track value in a polar plot:

1. Polar coordinates: This is the default setting. The magnitude and phase mouse pointer location are displayed
2. Rectangular coordinates: The real and imaginary part of the mouse pointer location are displayed.
3. Z: This is intended for use with s11 and s22 (reflection coefficients). You can use it on general polar plots--you'll just get negative resistance values outside of the unity reflection coefficient circle. The impedance at the mouse location on the smith plot is displayed.
4. Y: The admittance at the mouse pointer location on the Smith plot is displayed.

x-Axis Menu

Fig. 5.7.15: x-Axis menu (of a plot window)

Log Scale (x-Axis)--ac rectangular plots only
These menu items just convert the x-axis of the plot to a log axis--they have no effect on the frequency spacing of the data points. Since all traces always share a single horizontal axis, the entire plot is redrawn. This operation is not allowed on either transient or polar plots.

Linear Scale (x-Axis)--ac rectangular plots only
These menu items just convert the x-axis of the plot to a linear axis--they have no effect on the frequency spacing of the data points. Since all traces always share a single horizontal axis, the entire plot is redrawn.

Set Limits (x-Axis)
This menu item causes a small dialog box to pop up allowing you to change the horizontal axis limits of the active trace. Since all traces always share a single horizontal axis, all traces will be rescaled. Note that in a polar plot, the x-axis corresponds to the real axis and the y-axis corresponds to the imaginary axis.

y-Axis Menu

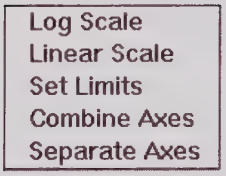


Fig. 5.7.16: y-Axis menu (of a plot window)

Log Scale (y-Axis)--ac rectangular plots only
These menu items just convert the y-axis of the active trace to a log axis--they have no effect on the data points. Any other trace that uses the same y-axis as the active trace will also be converted. This operation is not allowed on either transient or polar plots.

Linear Scale (y-Axis)--ac rectangular plots only
These menu items convert the y-axis of the active trace to a linear scale--they have no effect on the data points. Any other trace that uses the same y-axis as the active trace will also be converted.

Set Limits (y-Axis)
This menu item causes a small dialog box to pop up allowing you to change the vertical axis limits of the active trace. Any other trace that uses the same y-axis as the active trace will also be rescaled. Note that in a polar plot, the x-axis corresponds to the real axis and the y-axis corresponds to the imaginary axis.

Combine Axes--rectangular plots only
All selected traces are combined onto a new y-axis. Other traces already using the original y-axis are unaffected.

Separate Axes--rectangular plots only
All selected traces are put on a new y-axis. The original axis used by each trace is deleted if no other traces are left on the original axis.

Graticule Menu

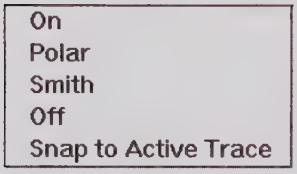


Fig. 5.7.17: Graticule menu (of a plot window)

(Graticule) On--rectangular plots only
This menu selection turns on the rectangular graticule in the plot window. This is just a grid of horizontal and vertical dashed lines drawn at nice intervals.

(Graticule) Off
This menu selection turns off the graticule in the plot window.

Polar--polar plots only
This menu selection turns on the polar graticule in the plot window. This graticule is composed of concentric circles centered at coordinates 0,0.

Smith--polar plots only
This menu selection turns on the Smith graticule. This graticule is composed of several equi-impedance circles and covers the polar plot on the real (horizontal) axis from -1 to +3 and on the imaginary (vertical) axis from -1 to +1. Selecting the "Scale to Unity Circle" menu item will give the more traditional scaling from -1 to +1 on both the real and imaginary axis.

Snap to Active Trace--rectangular plots only
This menu selection causes the graticule to be associated with the axis of the active trace--in other words the graticule snaps to the active trace. This selection only has meaning when there are at least two axis on the plot (a common example of this is when magnitude and phase are both displayed in the plot window.)

Error Messages

Circuit Error: Circuit hierarchy may not be deeper than 20 layers.
Your project's circuit hierarchy is deeper than the maximum number of layers allowed by this version of TRAVIS--i.e., you have nested subcircuits within subcircuits deeper than the maximum number of allowed layers. This could also be an indication that you created a recursive circuit hierarchy--you would have received a warning to this effect at the time you tried to place a subcircuit that would have created a recursive hierarchy. You will not be able to run an analysis until this problem is fixed.

Circuit Error: Entire circuit hierarchy may not contain more than 10,000 primitives.
Your project's circuit hierarchy contains more than the maximum number of primitives allowed by this version of TRAVIS.

Circuit Error: Entire circuit hierarchy may not contain more than 10,000 transmission lines.
Your project's circuit hierarchy contains more than the maximum number of transmission lines allowed by this version of TRAVIS.

Circuit Error: Entire circuit hierarchy may not contain more than 15,000 components + node markers.
Your project's circuit hierarchy contains more than the maximum number of components + node markers allowed by TRAVIS.

Circuit Error: Need to enter circuit first.
In order to run a simulation, you must first enter circuitry into the "Main Circuit Window". See topics on "Placing Components and Wires" and "Editing the Circuit".

Circuit Error: Total number of nodes in the entire circuit hierarchy may not exceed 200.
Your project's circuit hierarchy contains more than the maximum number of nodes allowed by this version of TRAVIS. You will not be able to run an analysis until this problem is fixed.

Circuit Error: A single circuit or subcircuit may not contain more than 1,000 components.
One of the subcircuits in the circuit hierarchy contains more than the maximum number allowed. This error appears at the instant you try to place more components than are allowed. This number includes primitives, node markers, and subcircuit components.

Circuit Error: A single circuit or subcircuit may not contain more than 15,000 wire segments.
One of the subcircuits in the circuit hierarchy contains more than the maximum number allowed. This error appears at the instant you try to place more wires or wire segments than are allowed.

Circuit Error: The component library may not contain more than 500 components.
You have attempted to have more than the maximum number of allowed components in the component library. If you want to add more subcircuit components to the library, you must remove some others first.

Circuit Error: Duplicate ports found.
TRAVIS found two ports with the same name in the circuit when a port analysis was run. Either rename or remove one of the ports. Another possible cause of this error is that there is a port placed within a subcircuit, and that subcircuit has multiple instances in the current circuit hierarchy. Since there are multiple instances of the subcircuit, there are multiple instances of everything in the subcircuit, including the port. For more information, see the section on "Port Analysis".

Circuit Error: No ports found.
TRAVIS found no ports with either the label "Port1" or "Port2" when a port analysis was run. Either no ports have been placed in the circuit or the ports that are placed have been given labels other than the two above. For more information, see the section on "Port Analysis".

Error: <component name> is the wrong type of component.
In the "depfunc" function found in dependent sources, you specify which meter it is dependent on. For example "v=depfunc(VM1,5)" means that the voltage of the dependent source will equal 5 times the voltage or current of the meter called VM1. However, if VM1 were, for example, a resistor, this error would be generated. The primitive that the dependent source depends on must be of the correct type. See information on specific TRAVIS dependent sources in the primitive component list for information on the allowed meter types.

Error: Component can't refer to itself.

In the "depfunc" function found in dependent sources, you specify which meter it is dependent on. A dependent source can not be dependent on its own signals--this error message is generated when a dependent source tries to refer to itself.

Error: Dependent source depends on nonexistent meter.
In the "depfunc" function found in dependent sources, you specify which other component it is dependent on. This error is generated if the element (meter) that it is dependent on cannot be found by TRAVIS anywhere else in the same circuit. Note that a dependent source can be dependent only on another meter within the same circuit, not on other meters elsewhere in the circuit hierarchy.

Error: Function has illegal number of parameters.
If a function was specified with the wrong number of parameters, this error will be generated. For example, "pulse(0,1,0)" is illegal because this function must have at least the first five parameters specified. This error may also be generated by other syntax errors that cause TRAVIS to interpret the function with too few parameters. For example, if you specified "pulse(0,1,2n,(4n+4n)),3n,10n,20n)", this error would be generated as well because the extra closing parenthesis signifies the end of the function specification to TRAVIS. See information on specific TRAVIS primitive components for information on proper function syntax.

Error: Function syntax error.
Functions such as "pulse(0,1,0,0,0,1n,2n)" or "depfunc(v1,5)" must have a certain format. Errors in this category occur when there is a missing parameter (e.g., pulse(0,,0,0,0,1n,2n)) or a parameter is longer than the maximum allowed 100 characters. See information on specific TRAVIS primitive components for information on proper function syntax.

Error: Illegal parameter value.
This error occurs when a component parameter is assigned to a value beyond its legal range. For example, the resistance of a resistor must be greater than or equal to zero. Another example of this error is when the tran=pulse parameters are impossible to realize, for example if the period is shorter than the pulse width. See information on specific TRAVIS primitive components for information on legal values for parameters.

Error: Required parameter is missing.
Primitive components must have certain parameters present in order for an analysis to be run. For example, a resistor must have the r=<resistance> parameter specified in its parameter list--if this is missing, then this error is generated. See information on specific TRAVIS primitive components for information on which parameters are required.

Error: Unknown function is specified.
You have specified a function that TRAVIS does not understand in the given context. Most likely you have made a typographical error in the function name.

Error: Unresolvable expression.
This error results when TRAVIS was unable to resolve an expression. "Expression" refers to anything from a single constant to a more complicated expression including variables, mathematical operations, and parentheses. The parameters in functions are also considered expressions in this context. TRAVIS attempts to add more specific error messages to this error message to help you figure out what the problem. Below is a list of examples of these specific error messages:

"...'x' is an undefined variable": The erroneous expression contains the variable "x". TRAVIS was unable to resolve "x" because you have not defined it.

"...syntax error occured at or near '#'": The erroneous expression contains a syntax error or something that TRAVIS does not understand--in this case a "#" character. TRAVIS will always indicate the object in the expression that is not following the rules of expression syntax.

"...syntax error in expression": This error will occur when there is a syntax error in an expression, but TRAVIS was unable to flag the specific violation.

"...mismatched parentheses": This error occurs when there are an unequal number of closing and opening parentheses in the expression.

"...illegal parenthesis placement": This means that TRAVIS encountered a parenthesis where it didn't expect one. For example "r=(5+)5".

"...can't specify a trace value here": You may not put trace values into component expressions. An example might be "i=v(15)/1k" in which v(15) refers to a trace.

"...parentheses nested too deeply--20 deep is the maximum": TRAVIS cannot interpret expressions with parenthesis nested more than 20 deep.

For more information see also: "Variables" and "Mathematical Expressions."

Analysis Error: Matrix is too big.
Matrix is too big. The circuit analysis matrix is larger than 4096 by 4096. The size of the circuit matrix depends on both the size of the circuit (number of nodes) and the connectivity of the circuit (number of circuit branches). You will need to simplify or reduce your circuit to complete an analysis.

Analysis Error: Insufficient memory for matrix.
The circuit matrix has grown too large to fit in available memory. Shrink the size of the circuit or free up additional memory to fix this error.

Analysis Error: Fatal: internal error #, please report to vendor.
Please write down the number of the error and report to the vendor.

Analysis Error: Cmatrix is singular...can not invert.
The ac analysis matrix is singular. Slight alterations in the circuit could help it converge (such as large resistors in parallel with metastable nodes to cause them to resolve to a certain state).

Analysis Error: Warning: matrix is near singular, accuracy questionable
The circuit matrix is near singular, which may result in numerical inaccuracies. This could be caused by nodes having a very high impedances to ground or loop impedances being very low.

Analysis Error: Fatal: system matrix is singular.
This error occurs when the circuit contains a voltage or current that can not be resolved. This can be caused by a loop in the circuit that contains only voltage sources, current meters, inductors, and transmission lines. This error is also caused when a node has no conductive path to ground or when you have forgotten to add ground to the circuit.

Analysis Error: Too many pulses, simulation truncated at #.
A single source may not generate more than 250 pulses over the course of the analysis. If it does, the analysis will be truncated.

Analysis Error: Too many sin cycles, simulation truncated at #
A single source may not generate more than 250 sinewave cycles over the course of the analysis. If it does, the analysis will be truncated.

Analysis Error: Fatal: dc convergence failed.
This can occur in complex circuits, circuits with feedback (especially positive feedback). Complex circuits are more susceptible to this error when there are diodes in the circuit.

Analysis Error: Warning: small pivot in matrix, results questionable.
When reordering the circuit matrix, no element was found which satisfies the absolute threshold criteria. The largest element in the matrix was chosen as the pivot. In lay-man's terms, this means that the matrix is probably near singular, which may result in numerical inaccuracies.

Analysis Error: Fatal: zero diagonal in matrix.
A zero was encountered on the diagonal of the matrix. This does not necessarily imply that the matrix is singular, however TRAVIS was unable to invert it. Deleting some wires and drawing them in a different order (to cause the automatic node numbering to occur differently) will reorder the matrix and might cure the problem.

Analysis Error: Fatal: out of memory.

The simulator ran out of memory during analysis. You will have to shrink the size of your circuit or free up more memory.

Analysis Error: Fatal: Ac bias convergence failed.
This can occur in complex circuits and circuits with feedback (especially positive feedback). Complex circuits are more susceptible to this error when there are diodes in the circuit.

Analysis Error: The forward and reverse bias regions of this diode overlap such that the simulator will be unable to solve the diode equations. Either 'BV' is too small or 'n' is too large.
The forward and reverse regions of operation of a diode are modeled separately. For any practical diode, these two regions are well separated. However, it is possible to specify a diode in TRAVIS in which these two regions overlap. Such a situation can occur if, for example, you set the breakdown voltage (BV) too low such that the diode starts to experience significant reverse breakdown current at the same time that it has forward current. This can also occur if the emission coefficient (n) is set very large. The simulator generates this error message if it will not be able to solve the diode equations due to this problem. To fix the problem, increase 'BV' or decrease 'n', or both.

Glossary

Active Plot Window
This is the plot window that all new traces are sent to when the "Use Active Plot" menu item is selected. If the "Open New Plot" menu item is selected, then a new plot window is opened when a new trace is created. This new plot window becomes the active plot.

Active Trace
This is the trace in the plot window whose name is displayed above the plot. It is also the trace that is used as the reference trace for the trace track value. There is only one active trace per plot. Whenever a trace is clicked with the left mouse button, it becomes the active trace.

Analysis Status Dialog Box
Any time an analysis is run (or the netlist is written out), the Analysis Status dialog box pops up to display the progress of the analysis. If there is an error in the circuit or the analysis, the analysis status dialog box will display the error message (you can also hit the "F1" key to pop up more detailed information on error messages).

If you want to terminate an analysis before it is completed, you can click the "Cancel" button. Any analysis results calculated up to that time are available for plotting, animating, etc.

Animation Plot
This is the small plot that sits on top of a transmission line component. When an animation is played, the waveforms on all visible animation plots will move as animation time passes. See "Transmission Line Analysis Results" for more information.

Circuit Window
This refers to an editing window in TRAVIS that contains a circuit. The two types of circuit windows are the main circuit window and the subcircuit window.

Component
Components are one of the building blocks of a circuit (wires and node markers are the others). There are two main types of components in TRAVIS: 1) primitives, and 2) subcircuit components.

Default Parameters
Refers to the parameters that a component has when it is placed from the "Place" menu or Subcircuit Library. Primitive component default parameters are hardwired into TRAVIS. For detailed information on the primitive component default parameters, see "Primitive Components". Subcircuit component default parameters are specified by the user and are edited in the Main Circuit Parameter dialog box.

Main Circuit
This refers to the top circuit in the circuit hierarchy of the project.

Main Circuit Window
This is the main window that opens up when TRAVIS first starts. The circuit that you edit in this window is called the main circuit. All file operations are done from this window.

Modes
The mode that the circuit editor is in determines what functions the mouse performs. The status bar displays the current mode. Mode names are usually just a simple description of what operations are currently available--especially mouse button operations. For example, when you are in "Edit circuit" mode, clicking the left mouse button on a component will cause a component editor dialog to pop up. In "Draw wire" mode, clicking the left mouse button causes a wire to be started or placed. In "Click component/wire for ac polar plot" mode, all signal traces are put into an ac polar plot. In most cases, clicking the right mouse button once or twice causes you to go back to "Edit circuit" mode.

Parameter Assignment
This refers to the expression that is entered to specify component parameters and their values. "r=1k" in a resistor is an example of a parameter assignment. In this example, "r" represents the component's resistance. This example will result in the resistance parameter having a value of 1k ohms. Another example is "label=T12". In this example, this component is given "T12" as a label. These are entered into the parameter field of the Edit Component dialog box.

Parameter Field
This is the bottom section of the Edit Component dialog box and the Edit Main Circuit Parameter dialog box. Parameter and variable assignments as well as comments are entered here.

Plot Window
This is the window that contains a plot of a waveform from an analysis. To see a plot from an analysis:

1. Select one of the following from the "See Results" menu:
   a) If you ran a transient analysis, select the "Transient Plot" menu item.
   b) If you ran an ac analysis, select either the "AC Plot->Rectangular" or "AC Plot->Polar" menu item.
   c) If you ran a port analysis, select either the "Port Plot->Polar" or "Port Plot->Rectangular" menu item.

2) Click on the wire or component whose voltage or current you want to see.

A plot window containing a plot of the waveform pops up.

Primitive Component
Primtive components (or primitives) are the basic components that are hardwired into the "Place" menu and are directly understood by TRAVIS (e.g., resistors, capacitors). For more information on primitives see Primitive Components.

Project
Refers to everything that is currently being edited. The project consists of the main circuit (including its component representation and default parameters), the Subcircuit Library, all of the analysis setup parameters, and the most recent analysis results.

Quickref Menu
This is the submenu that is found under the "Place->Subcct Component Quickref" menu item. It is a quick access list of subcircuit components. This menu is user configurable (see Subcircuit Library).

Resolved Information Field
This is the top section of the Edit Component dialog box and the Edit Main Circuit Parameter dialog box. You cannot edit the information in this field--it displays the final values of variables and parameters from the "Parameter Field". It also displays the type of component and the node numbers (or names) that are connected to the component's terminals.

Selected Trace
Traces on a plot that have been clicked on with the left mouse button. A trace can also be selected by clicking on its label. Selected traces are drawn red. There can be multiple selected traces in a plot, whereas there can be only one active trace.

Status Text
Refers to the text located in the upper left corner of the circuit editor window just under the menu bar (near the "File" menu item). The status text displays what mode the circuit editor is currently in and thus often shows what you are supposed to do next. An exception to this is if you are tracking a animation plot. In this case the status bar will contain the trace track value.

Subcircuit
Refers to a circuit that can be or is used within another circuit.

Subcircuits are created when the main circuit of the project is saved as a subcircuit to disk. A subcircuit file is created that contains the subcircuit and its body (the subcircuit component). A subcircuit can later be individually loaded and edited (by selecting "Open Subcircuit" from the "File" menu) or loaded into a project (by selecting "New Subcct Component from Disk" from the "Place" menu). When a subcircuit is loaded into a project, it is put in that project's Subcircuit Library.

When a subcircuit is placed into a project circuit, it is represented as a subcircuit component.

Subcircuit Component
This is a component that contains a subcircuit. Subcircuit components look and, in many ways, act just like primitive components. The subcircuit that is represented by (contained within) the subcircuit component may in turn contain primitives or other subcircuit components. The result is a hierarchical circuit structure. At the bottom of this hierarchy, the circuit is ultimately composed entirely of primitive components.

An example of a subcircuit component might be an op-amp that would contain a subcircuit composed of several primitive components (such as dependent sources, resistors, and capacitors).

Subcircuit Window
When you edit a subcircuit that is already placed in the current circuit, a special circuit window called the subcircuit window pops up. The only difference between this window and the main circuit window is that the file operation menu items are missing along with various other menu items not pertinent to a placed subcircuit. See "Editing Subcircuits Already Placed in a Circuit" for more information.

Trace Tracking
You can do trace tracking on any TRAVIS plot. Simply activate the window that the plot is in (click the title bar or body of the window), then move the mouse pointer over the plot. When you do this, the trace track value starts indicating the location in the plot of your mouse pointer. If you then hit the arrow (cursor) keys, the mouse pointer (which is a cross hair) will snap to the active trace and display the coordinates of the current trace point in the upper left corner of the plot window. In the case of animation plots, the trace track value will appear in the status bar. In plot windows orignating from the Transmission Line Designer Tool, a status message indicating the validity of the current trace point is displayed under the trace track value.

Trace Track Value
Refers to the coordinates displayed in the upper left corner of the plot window. The coordinates respresent the location of the mouse pointer on the plot. For animation plots, the trace track value is displayed in the status bar and includes a label to indicate which signal is being tracked.

TRAVIS Clipboard

This is the internal clipboard that TRAVIS uses for copying and pasting circuitry. This is isolated from the Windows clipboard, which is affected only by the "Send Window to Clipboard" function under the "File" menu of the circuit window or the plot window.

Index

AC Analysis, 49
Analysis, 49
    AC Analysis, 49
        Polar Plots, 77
        Rectangular Plots, 76
        Running, 56
        Seeing Results, 59
        Setting up, 51
        Waveform Math, 76, 77
    AC Animation Plots
        Setting up, 67
    Analysis Menu, 51
    DC Analysis, 49
        DC Math, 80
        Removing Voltages, 81
        Running, 56
        Seeing Results, 59
        Showing voltages, 80
    Plots
        Bring Plots to Top, 65
        Opening a New Plot, 65
        Sticking Plot Windows on Top, 65
        Traces Using Existing Plot Axis, 66
        Using an Existing Plot, 65
    Port Analysis, 50
        Polar Plots, 78
        Ports, 19
        Rectangular Plots, 79
        Running, 57
        Seeing Results, 60
        Waveform Math, 78, 79
    Running
        Analysis Status Dialog Box, 95

See Results Menu, 64
Seeing Results, 59
Transient Analysis, 51
Plots, 75
Running, 57
Seeing Results, 60
Setting up, 53
Waveform Math, 75
Transient Animation Plots
Setting up, 66
Transmission Line Analysis
AC Animation Control, 73
Animating AC Plots, 72
Animating Plots, 61
Animating Transient Plots, 69
Animation Speed, 70, 73
Seeing Results, 61
Setting up, 53
Trace Tracking Transmission Line Animation Plots, 72, 74
Transient Animation Control, 70
Zoom, 71, 74
Waveform Math, 63
Analysis Status Dialog Box, 95
Animating AC Animation Plots, 72
Step by step, 62
Animating Transient Animation Plots, 69
Step by step, 61
Body, 13
Circuit Window, 95
Modes, 96
Status Text, 97
Circuits
Editing, 34
Entering and Editing, 5
Placing Components, 17
Placing Wires, 18
Variables, 24
Clipboard, 8, 99
Components, 20, 95
Default Parameters, 96
Labels, 21
Moving, 23
Parameter Assignment, 96
Parameter Field, 96

Parameters, 20
Commenting out, 22
Syntax, 22
Placing, 17
Primitive Components, 25, 97
Capacitor, 25
Current Meter, 33
Current Source, 28
Dependent Current Source, 32
Dependent Voltage Source, 32
Diode, 29
Inductor, 26
Resistor, 25
Transmission Line, 31
Volt Meter, 33
Voltage Source, 26
Resolved Information Field, 97
Subcircuit Component, 98
DC Analysis, 49
Deleting Files, 6
Drawing Wires, 18
Edit Menu, 10
Editing
Circuit, 34
Component Labels, 21
Moving, 23
Component Parameters, 20
Components
Parameter Field Syntax, 22
Copy, 12
Cut, 12
Cut Whatever Is Touched, 12
Edit Menu, 10
Move, 13
Paste, 12
Placing a New Subcircuit from Disk, 38
Placing a Previously Loaded Subcircuit, 39
Placing Components, 17
Placing Ground/Nodes Markers/Terminals/Ports, 19
Placing Subcircuit Components, 38
Placing Wires, 18
Redo, 11
Subcircuit Body, 36
Subcircuit body, 13

Subcircuit Terminals, 36
Subcircuits, 34, 35
Subcircuits Placed in the Circuit, 40
Undo, 11
Zoom, 14
Editing Component Parameters, 20
Editing the Subcircuit Body, 36
Entering and Editing a Circuit, 5
Error Messages, 89
Exporting schematics from TRAVIS to other applications, 8
File Menu, 5
File Operations
Deleting Project/Circuit Files, 6
Loading a Project, 6
Loading a Subcircuit, 8
Renaming Project/Circuit Files, 6
Saving the Project, 7
Saving the Subcircuit, 8
Glossary, 95
Grounds, 19
Labeling Nodes (Node Markers), 19
Labels, 21
Loading a Project, 6
Loading a Subcircuit, 8
Main Circuit, 96
Main Circuit Parameters, 15
Main Circuit Window, 96
Mathematical Expressions, 23, 24
Menus
Circuit Window
Analysis, 51
Edit, 10
File, 5
Place, 17
See Results, 64
Tools
Transmission Line Designer, 42
Plot Window, 83
Quickref Menu, 97
Subcircuit Body Window, 37
Transmission Line Designer, 44
Netlist, 58
Node Markers, 19
Opening a Project, 6

Opening a Subcircuit, 8
Place Menu, 17
Placing Components, 17
Placing Subcircuit Components, 38
Placing Wires, 18
Plot Windows, 81
    Writing a Trace to a File, 84
Plots, 81
    AC Analysis, 76, 77
    Active Plot Window, 95
    Active Trace, 81, 95
    Animation Plot, 95
    Autoscaling a Plot, 85
    Axis
        Combining multiple traces onto a single axis, 87
        Horizontal Linear Scale, 86
        Horizontal Log Scale, 86
        New Traces Use Exisiting, 85
        Separating multiple traces onto separate axes, 87
        Set Horizontal Limits, 86
        Setting Vertical Limits, 87
        Vertical Linear Scale, 87
        Vertical Log Scale, 87
    Deleting Traces from a Plot, 85
    Graticule, 82
        Changing Traces, 88
        Polar, 88
        Rectangular, 87
        Smith Chart, 88
        Turn Off, 87
    Page Setup, 84
    Placing Animation Plots, 80
    Plot Axis, 81
    Plot Window, 96
    Plot Window Menu, 83
    Port Analysis, 78, 79
    Printer Setup, 84
    Printing, 84
    Removing Animation Plots, 81
    Scaling Polar Plots to Smith Chart Limits, 86
    Selected Trace, 97
    Sending Plot to Windows Clipboard, 84
    Sticking Plot Window on Top, 85
    Trace Labels, 82

Trace Track Value, 98
Trace Tracking, 82, 98
Trace Tracking Polar Plots, 86
Track Value, 81
Transfer Plot, 82
Transient Analysis, 75
Turning on/off Trace Dots, 85
Turning on/off Trace Labels, 85
Writing a Trace to a File, 84
Zooming in and out, 82
Port Analysis, 50
Ports, 19
Printing
Page Setup, 9
Printer Setup, 10, 84
Printing the Circuit, 9
Printing Plots, 84
Page Setup, 84
Printer Setup, 84
Project, 97
Loading from Disk, 6
New, 6
Saving to Disk, 7
Pulse Transient Function, 26
Quickref Menu, 39, 97
Editing, 40
Redoing Editing Operations, 11
Refreshing the Circuit Window, 11
Renaming Files, 6
Resolved Information Field, 97
Saving a Subcircuit, 8
Saving the Project, 7
Sin Transient Function, 26
Status Text, 97
Subcircuit, 98
Subcircuit Body, 13
Subcircuit Library, 15, 40
Subcircuit Library Dialog Box, 40
Subcircuit Window, 98
Subcircuits, 34
Body, 13
Creating and Editing, 35
Editing Subcircuits Already Placed in the Circuit, 40
Editing the Subcircuit Body, 36

Menu in Body Editor Window, 37
Editing the Subcircuit Parameters, 15
Getting a New Subcircuit from Disk, 38
Loading from Disk, 8
Parameters, 35
Placing a Previously Loaded Subcircuit, 39
Placing in Another Circuit
Loading from Disk, 38
Loading from QuickRef, 38
Placing in the Circuit, 38
Saving to Disk, 8
Subcircuit Library, 40
Subcircuit Library Dialog Box, 40
Subcircuit Library Menu Item, 15
Terminals, 19, 36
Variables, 24
Terminals, 19
Transient Analysis, 51
Transmission Line Designer, 13, 42
Transmission Lines
Animating AC Plots, 72
Animating Plots, 61
Animating Transient Plots, 69
Placing Plots, 80
Primitive Component, 31
Removing Plots, 81
Transmission Line Designer, 13, 42
TRAVIS Clipboard, 99
Undoing Editing Operations, 11
Units, 42
Variables, 24
View Scale, 14
Waveform Math, 63, 75, 76, 77, 78, 79
Wires, 18
Zoom, 14

The Artech House Microwave Library

Acoustic Charge Transport: Device Technology and Applications, R. Miller, C. Nothnick, and D. Bailey

Advanced Automated Smith Chart Software and User's Manual, Version 2.0, Leonard M. Schwab

Algorithms for Computer-Aided Design of Linear Microwave Circuits, Stanislaw Rosloniec

Analysis, Design, and Applications of Fin Lines, Bharathi Bhat and Shiban K. Koul

Analysis Methods for Electromagnetic Wave Problems, Eikichi Yamashita, editor

Automated Smith Chart Software and User's Manual, Leonard M. Schwab

C/NL2 for Windows: Linear and Nonlinear Microwave Circuit Analysis and Optimization, Software and User's Manual, Stephen A. Maas and Arthur Nichols

Capacitance, Inductance, and Crosstalk Analysis, Charles S. Walker

Design of Impedance-Matching Networks for RF and Microwave Amplifiers, Pieter L. D. Abrie

Dielectric Materials and Applications, Arthur von Hippel, editor

Dielectrics and Waves, Arthur von Hippel

Digital Microwave Receivers, James B. Tsui

Electric Filters, Martin Hasler and Jacques Neirynck

Electrical and Thermal Characterization of MESFETs, HEMTs, and HBTs, Robert Anholt

E-Plane Integrated Circuits, P. Bhartia and P. Pramanick, editors

Feedback Maximization, Boris J. Lurie

Filters with Helical and Folded Helical Resonators, Peter Vizmuller

Frequency Synthesizer Design Toolkit Software and User's Manual, Version 1.0, James A. Crawford

Fundamentals of Distributed Amplification, Thomas T. Y. Wong

GaAs MESFET Circuit Design, Robert A. Soares, editor

GASMAP: Gallium Arsenide Model Analysis Program, J. Michael Golio *et al.*

Handbook of Microwave Integrated Circuits, Reinmut K. Hoffmann

Handbook for the Mechanical Tolerancing of Waveguide Components, W. B. W. Alison

HELENA: HEMT Electrical Properties and Noise Analysis Software and User's Manual, Henri Happy and Alain Cappy

HEMTs and HBTs: Devices, Fabrication, and Circuits, Fazal Ali, Aditya Gupta, and Inder Bahl, editors

High-Power Microwave Sources, Victor Granatstein and Igor Alexeff, edtiors

High-Power GaAs FET Amplifiers, John Walker, editor

High-Power Microwaves, James Benford and John Swegle

Introduction to Microwaves, Fred E. Gardiol

Introduction to Computer Methods for Microwave Circuit Analysis and Design, Janusz A. Dobrowolski

Introduction to the Uniform Geometrical Theory of Diffraction, D. A. McNamara, C. W. I. Pistorius and J. A. G. Malherbe

LOSLIN: Lossy Line Calculation Software and User's Manual, Fred E. Gardiol

Lossy Transmission Lines, Fred E. Gardiol

Low-Angle Microwave Propagation: Physics and Modeling, Adolf Giger

Low Phase Noise Microwave Oscillator Design, Robert G. Rogers

MATCHNET: Microwave Matching Networks Synthesis, Stephen V. Sussman-Fort

Matrix Parameters for Multiconductor Transmission Lines: Software and User's Manual, A. R. Djordjevic *et al.*

Microelectronic Reliability, Volume I: Reliability, Test, and Diagnostics, Edward B. Hakim, editor

Microelectronic Reliability, Volume II: Integrity Assessment and Assurance, Emiliano Pollino, editor

Microwave and Millimeter-Wave Diode Frequency Multipliers, Marek T. Faber,
Jerzy Chamiec, Miroslaw E. Adamski

Microwave and RF Circuits: Analysis, Synthesis, and Design, Max Medley

Microwave and RF Component and Subsystem Manufacturing Technology, Heriot-Watt University

Microwave Circulator Design, Douglas K. Linkhart

Microwave Engineers' Handbook, 2 Volumes, Theodore Saad, editor

Microwave Materials and Fabrication Techniques, Second Edition, Thomas S. Laverghetta

Microwave MESFETs and HEMTs, J. Michael Golio *et al.*

Microwave and Millimeter Wave Heterostructure Transistors and Applicatons, F. Ali, editor

Microwave and Millimeter Wave Phase Shifters, Volume I: Dielectric and Ferrite Phase Shifters, S. Koul and B. Bhat

Microwave and Millimeter Wave Phase Shifters, Volume II: Semiconductor and Delay Line Phase Shifters, S. Koul and B. Bhat

Microwave Mixers, Second Edition, Stephen Maas

Microwave Transmission Design Data, Theodore Moreno

Microwave Transition Design, Jamal S. Izadian and Shahin M. Izadian

Microwave Transmission Line Couplers, J. A. G. Malherbe

Microwave Tubes, A. S. Gilmour, Jr.

Microwaves: Industrial, Scientific, and Medical Applications, J. Thuery

Microwaves Made Simple: Principles and Applicatons, Stephen W. Cheung, Frederick H. Levien *et al.*

MMIC Design: GaAs FETs and HEMTs, Peter H. Ladbrooke

Modern GaAs Processing Techniques, Ralph Williams

Modern Microwave Measurements and Techniques, Thomas S. Laverghetta

Monolithic Microwave Integrated Circuits: Technology and Design, Ravender Goyal *et al.*

Nonuniform Line Microstrip Directional Couplers, Sener Uysal

PC Filter: Electronic Filter Design Software and User's Guide, Michael G. Ellis, Sr.

PLL: Linear Phase-Locked Loop Control Systems Analysis Software and User's Manual, Eric L. Unruh

RF Design Guide: Systems, Circuits, and Equations, Peter Vizmuller

Scattering Parameters of Microwave Networks with Multiconductor Transmission Lines: Software & User's Manual, A. R. Djordjevic *et al.*

Solid-State Microwave Power Oscillator Design, Eric Holzman and Ralston Robertson

Terrestrial Digital Microwave Communications, Ferdo Ivanek *et al.*

Time-Domain Response of Multiconductor Transmission Lines: Software and User's Manual, A. R. Djordjevic *et al.*

Transmission Line Design Handbook, Brian C. Waddell

TRAVIS Pro: Transmission Line Visualization Software and User's Manual, Professional Version, Robert G. Kaires and Barton T. Hickman

TRAVIS Student: Transmission Line Visualization Software and User's Manual, Student Version, Robert G. Kaires and Barton T. Hickman

Yield and Reliability in Microwave Circuit and System Design, Michael Meehan and John Purviance

For further information on these and other Artech House titles, contact:

Artech House
685 Canton Street
Norwood, MA 02062
617-769-9750
Fax: 617-769-6334
Telex: 951-659
email: artech@world.std.com

Artech House
Portland House, Stag Place
London SW1E 5XA England
+44 (0) 171-973-8077
Fax: +44 (0) 171-630-0166
Telex: 951-659
bookco@artech.demon.co.uk

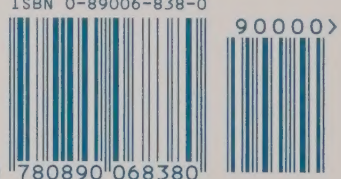
◁ BREAK AT SEAL AND PULL TO OPEN ▷
PATENT 4,549,658 #31-FD PATENT D-286,468

TRAVIS Student

Transmission Line Visualization Software and User's Manual

Student Version

ISBN 0-89006-838-0



Artech House Publishers BOSTON • LONDON